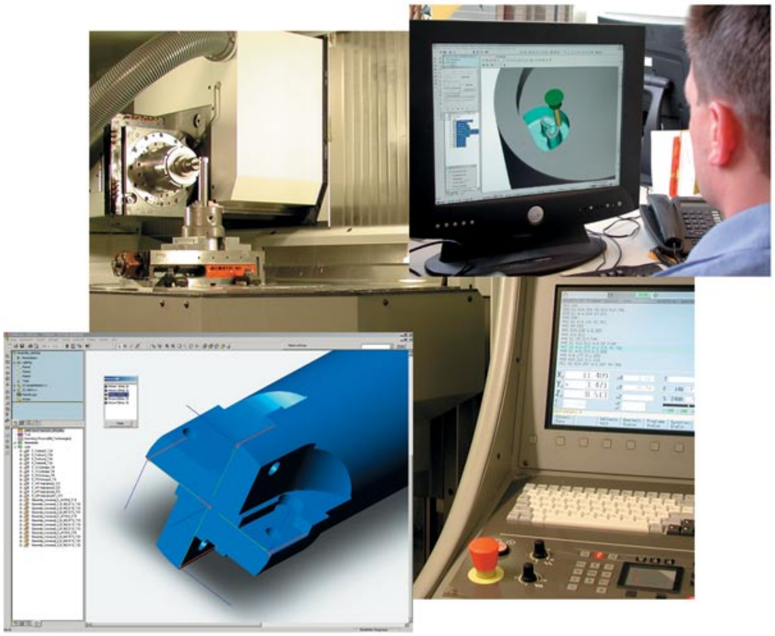
VISIT…

LANZAROTE
Caliente.COM

SolidCAM

The integrated CAM-Engine for SolidWorks



SolidCAM Milling User's Guide

Volume #1

SolidWorks Certified Gold Product

©1995-2004 SolidCAM LTD.

All Rights Reserved.

Contents

Volume #1

1. SolidCAM Basics

1.1 Installing SolidCAM software....14

- 1.1.1 System requirements....14
- 1.1.2 SolidCAM Single License Installation....14
- 1.1.3 SolidCAM Network License Installation....19
- 1.1.4 SolidCAM Dongle Update....21

1.2 Basics Concepts....22

1.3 Starting SolidCAM....22

1.4 SolidCAM Interface....23

1.5 Getting Help....25

2. CAM-Part

2.1 Starting a new Milling CAM-Part....28

- 2.1.1 The structure of the CAM-Part....30

2.2 Defining Home....31

- 2.2.1 Multi-Sided Home....31
- 2.2.2 Define Home....32
- 2.2.3 Add Home within Multi-Sided Mode....33
- 2.2.4 Defining the homes for 3-axis CNC machine within Multi-Sided mode....39
- 2.2.5 Defining the homes for 4-axis CNC machine within Multi-Sided mode....40
- 2.2.6 Defining the homes for 5-axis CNC machine within Multi-Sided mode....42
- 2.2.7 Home definition methods....44
- 2.2.8 Add Home within Projections Mode....46
- 2.2.9 Defining the homes for 3-axis CNC machine within Projected mode....47
- 2.2.10 Defining the homes for 4-axis CNC machine within Projected mode....49
- 2.2.11 Defining the homes for 5-axis CNC machine within Projected mode....51
- 2.2.12 Home data dialog....53
- 2.2.13 SolidCAM coordinate system....55

2.3 Home operations....56

2.4 Stock and Target Model....58

2.4.1 Target model definition....60
2.4.2 Stock Model....61
2.4.3 Updating Stock Model....65

2.5 Tool options....67

2.6 Mac Options....69

2.7 Work Material....70

2.8 CNC-Controller and Axis type....71

2.8.1 Axis type....71

2.9 Default G-Code numbers....74

2.10 Managing CAM-Parts....76

2.11 CAM-Part Documentation....81

2.11.1 Documentation Editor....82

3. Tools

3.1 User-defined Tool Types....86

3.1.1 Add new Tool Types....87
3.1.2 Delete Tool Types....88

3.2 Tool Tables....89

3.3 Working with Part Tool Table....90

3.4 Working with Current Tool Table....91

3.5 Managing tool tables....93

3.5.1 Edit Tool table....93
3.5.2 Create Tool table....94
3.5.3 Copy tool table....95
3.5.4 Delete tool table....96

3.6 Tool Table dialog....97

3.6.1 Tools Filter....98
3.6.2 Tool Range....99
3.6.3 Show Tool....100

3.6.4 Tool table - Edit ... 102
3.7 Managing tools ... 103
3.8 Tool data ... 108
3.8.1 Tool Type ... 109
3.8.2 Topology page ... 110
3.8.3 Default tool data / Job Data ... 115
3.9 Tool holding system ... 117
3.9.1 Tool holders dialog ... 118
3.9.2 Tool Holder Geometry definition ... 123
3.9.3 Tool Holder segments ... 124
3.9.4 Using Tool Holders ... 126
3.10 Shaped Tools ... 130
3.10.1 Shape-Tools dialog ... 131
3.10.2 Shape-tool geometry definition ... 135
3.10.3 Shaped Tool segments ... 136
3.10.4 Using Shaped tools ... 138
3.11 Feed and Speed default ... 142

4. Geometry

4.1 Introduction ... 148
4.2 Adding a new Geometry to your CAM-Part ... 149
4.3 Drill Geometry ... 150
4.3.1 Defining a Drill Geometry ... 150
4.3.2 Edit a Drill Geometry ... 151
4.3.3 Drill Geometry Selection Dialog ... 151
4.4 3D Model Geometry ... 156
4.4.1 Defining a 3D Model Geometry ... 156
4.4.2 Editing a 3D Model Geometry ... 158
4.4.3 CAD selection ... 159
4.5 Wireframe Geometry ... 160
4.5.1 Defining a Profile/Pocket Geometry ... 161

4.5.2 Defining a Working Area 162
4.5.3 Defining a Slot Geometry 164
4.5.4 Defining a Section Geometry 165
4.5.5 Defining a Limit Geometry 167
4.6 Editing Chain Geometries 169
4.6.1 Add Multi-chain 169
4.6.2 Adding a Chain 169
4.6.3 Chain operations 170
4.6.4 Settings 171
4.6.5 Spline approximation 171
4.7 Chain selection 172
4.7.1 Chain Options dialog 172
4.7.2 Chain selection dialog 179
4.7.3 Chains direction 181
4.7.4 Chains sorting 183
4.8 Operations with Geometries 186
4.8.1 Synchronization of the Wireframe geometry 187
4.8.2 Synchronizaton of the 3D Model geometry 192

5. Jobs

5.1 Adding a Job 196
5.2 SolidCAM Job Interface 197
5.2.1 Home 197
5.2.2 Geometry 198
5.2.3 Tool 199
5.2.4 Job name 203
5.2.5 Milling Levels 204
5.2.6 Extra parameters 208
5.2.7 Mirror/Rotate point 208
5.2.8 Message 209
5.2.9 Job operation buttons 209
5.3 Working with Jobs 211
5.3.1 Add Job 212

5.3.2 Add Machining process ... 213
5.3.3 Edit ... 213
5.3.4 Calculate/Calculate All ... 213
5.3.5 G-Code/G-Code All ... 213
5.3.6 Calculate & G-Code All ... 213
5.3.7 Simulate ... 213
5.3.8 File ... 214
5.3.9 Job Group ... 214
5.3.10 Delete/Delete all ... 215
5.4 Managing Jobs in the CAM-Manager tree ... 216
5.4.1 Job Sequence ... 216
5.4.2 Undo Sequence ... 216
5.4.3 Split ... 216
5.4.4 Expand ... 217
5.4.5 Collapse ... 217
5.4.6 Automatic Sort ... 218
5.5 Job Transformations ... 219
5.5.1 Init ... 220
5.5.2 Move ... 220
5.5.3 Translate ... 220
5.5.4 Rotate ... 222
5.5.5 Mirror ... 223
5.5.6 4th Axis ... 224
5.6 Fixture ... 225
5.6.1 Fixture dialog ... 226

6. 2.5D Milling

6.1 Profile Job ... 230
6.1.1 Geometry name ... 231
6.1.2 Down Step ... 232
6.1.3 Offsets ... 232
6.1.4 Fillet size for last cut ... 234
6.1.5 Define depth ... 235

6.1.6 Descent type 237
6.1.7 Rest Material/Chamfer 237
6.1.8 Tool Side 243
6.1.9 Compensation 244
6.1.10 Profile Direction 246
6.1.11 Profile extension 246
6.1.12 Approach/Retreat 246
6.1.13 Trochoidal milling 249
6.2 Pocket Job 250
6.2.1 Geometry name 251
6.2.2 Down Step 251
6.2.3 Pocket Types 252
6.2.4 Hatch type 253
6.2.5 Contour type 258
6.2.6 Hatch+Finish type 266
6.2.7 Clear type 267
6.2.8 Plunging Pattern 268
6.2.9 Overlap 270
6.2.10 Compensation 270
6.2.11 Offsets 270
6.2.12 Finish 271
6.2.13 Fillet size for last cut 272
6.2.14 Rest Material/Chamfer 272
6.2.15 Approach 272
6.2.16 Retreat 276
6.3 Drill Job 279
6.3.1 Geometry name 280
6.3.2 Tool 280
6.3.3 Drill depth 280
6.3.4 Drill cycle type 281
6.3.5 Sequence of drill positions 284
6.4 Slot Job 286
6.4.1 Geometry name 287
6.4.2 Depth type 287

6.4.3 Constant Depth type 287
6.4.4 Section depth type 289
6.5 Translated Surface Job 293
6.5.1 Geometry name 294
6.5.2 Define Profile Start Point 295
6.5.3 Profile direction 295
6.5.4 Tool side 295
6.5.5 Geometry on Section 296
6.5.6 Constraint 297
6.5.7 Process type 298
6.5.8 Surface Data for Translated Surfaces 298
6.5.9 Engraving Data for Translated Surface Job 301
6.5.10 Limits 302
6.5.11 Limit Data for Translated Surface 303
6.5.12 Engraving Limit for Translated Surface Job 306
6.5.13 Approach/Retreat 308
6.5.14 Show surface 308

7. 3D Milling

7.1 3D Model Job Overview 310
7.2 Working Area 311
7.2.1 Working area dialog 312
7.2.2 Working area definition via Geometry 313
7.2.3 Working area definition via Surface condition 316
7.2.4 Working area definition via Face selection 317
7.2.5 Working area definition via Rest material 319
7.3 Tolerance 321
7.3.1 Surface tolerance 322
7.3.2 Tool path tolerance 323
7.4 Roughing 324
7.4.1 Overlap 325
7.4.2 Down step 325
7.4.3 Clean flat 325

7.4.4 Offset 326
7.4.5 Fillet size for last cut 327
7.4.7.Mode (Open Pocket machining) 334
7.4.8 Rest Material 337
7.4.9 Hatch Roughing 340
7.4.10 Contour Roughing 341
7.4.11 Plunging pattern 342
7.5 Semi-Finish/Finish 343
7.5.1 Overview 343
7.5.2 Descent 346
7.5.3 Semi-Finish/Finish strategies 349
7.5.4 Linear finish strategy 350
7.5.5 Offset cutting Finish Strategy 356
7.5.6 Spiral Finish Strategy 361
7.5.7 Circular Pocket Finish Strategy 368
7.5.8 Constant-Z Finish Strategy 370
7.5.9 Pencil milling 380
7.6 3D Engraving Job 382
7.6.1 Geometry name 383
7.6.2 Engraving depth 383
7.6.3 Tolerance 383
7.6.4 Engraving geometry name 384
7.6.5 Engraving technology 384
7.7 3D Drill Job 387
7.7.1 Geometry name 388
7.7.2 Tool 388
7.7.3 3D Model Geometry name 388
7.7.4 Offset from model 389
7.7.5 Drill cycle type 389
7.7.6 Sequence of drill positions 390

8. Simulation

8.1 Introduction 392

8.1.1 Simulation modes ... 393
8.1.2 Simulation Controls ... 393
8.2 2D simulation mode ... 394
8.2.1 Simulation Control ... 394
8.2.2 Simulation toolbar ... 394
8.2.3 Show Data ... 395
8.2.4 Projection ... 395
8.2.5 Show tool ... 397
8.2.6 Erase on Z change ... 397
8.2.7 Stop on Next ... 397
8.2.8 Clear ... 398
8.2.9 Colors ... 398
8.2.10 Simulation speed ... 398
8.3 VerifyPlus simulation mode ... 399
8.3.1 Simulation toolbar ... 399
8.3.2 Buttons available in the Setup Mode ... 401
8.3.3 Buttons available in the Simulation Mode ... 403
8.4 Host CAD simulation mode ... 404
8.4.1 Simulation Control ... 404
8.4.2 Show Data ... 405
8.4.3 Show Tool ... 405
8.4.4 Stop on Next ... 405
8.4.5 Clear ... 405
8.4.6 Colors ... 405
8.4.7 Simulation speed ... 405
8.5 3D simulation mode ... 406
8.5.1 Simulation toolbar ... 406
8.5.2 Simulation control ... 408
8.5.3 Show Data ... 408
8.5.4 Stop on Next ... 408
8.5.5 Colors ... 408
8.5.6 Simulation speed. ... 408
8.6 SolidVerify simulation mode ... 409

8.6.1 Simulation Control 409
8.6.2 Show Data 409
8.6.3 Stop on Next 410
8.6.4 Clear 410
8.6.5 Simulation speed 410
8.6.6 Single color/Color by tool 410
8.6.7 Simulation toolbar 411
8.6.8 Simulation menu 417
8.7 Rest Material simulation mode 420
8.7.1 Simulation toolbar 420
8.7.2 Simulation control 422
8.7.3 Tolerance 423

9. G-Code

9.1 Generate 426
9.2 List 427
9.3 Copy 428
9.4 Print 428

10. SolidCAM Settings

10.1 Introduction 430
10.2 User Directories Settings 431
10.2.1 User Directory for SolidCAM-Parts 431
10.2.2 User Directory for SolidCAM-Tables 432
10.3 Units Settings 433
10.3.1 Metric/Inch 433
10.3.2 Approximation 434
10.3.3 Chain selection 434
10.3.4 Rest Material Calculation 435
10.3.5 Fillet size for last cut 435
10.4 Default CNC-controller settings 436
10.4.1 Post-processor files directory 436

  10.4.2 CNC-controllers 437
10.5 Color Setup 438
10.6 Auto-Save settings 440
10.7 External program settings 441
10.8 Editors Settings 442
10.9 DNC settings 443
10.10 Simulation Settings 444
10.11 Synchronization settings 445
10.12 Default Geometry Names 447
10.13 Rest Materal & SolidVerify Settings 448
10.14 Tool Settings 450
Index 453

SolidCAM Basics

1

SolidCAM is a powerful CAM product that has been especially designed for the shop floor. SolidCAM supports the complete range of major manufacturing applications including 2.5D Milling, 3D-Milling, Turning, Turning with Driven tools and Wire EDM, in one integrated solution. Machining operations can be defined on 2D Design Drawings as well as on 3D Solid and Surface Models. SolidCAM has also a powerful general post-processor tool that enables the easy customization of the G-Code file output to various types of CNC controllers

This chapter discusses some basic concepts and terminology used throughout SolidCAM.

❒ Installing SolidCAM
❒ Starting SolidCAM
❒ SolidCAM Basic concepts
❒ SolidCAM Interface
❒ Getting Help

1.1 Installing SolidCAM software

1.1.1 System requirements

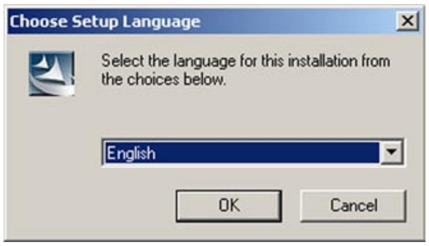
- Microsoft® Windows XP Professional (recommended) with Service Pack 1, Windows 2000 (recommended) with Service Pack 3 or 4; also runs on Windows NT with Service Pack 6.
- Intel Pentium®- or AMD Athlon™-class processor
- 256 MB RAM or more (512 MB or more recommended for large CAM-Parts machining)
- Graphics adapter (64MB RAM recommended)
- Pointing device
- CD-ROM drive
- Internet Explorer version 5.5 or later recommended

1.1.2 SolidCAM Single License Installation

1. Insert the SolidCAM Installation CD into the computer's CD-ROM drive; the installation will be started automatically.

You can also start the installation by running the setup.exe file. This file is located in the **SolidCAMXXXX/Disk1** folder on the SolidCAM Installation CD. (**XXXX** is the current version of SolidCAM)

2. The **Choose Setup Language** dialog will be displayed.



Choose Setup Language

Select the language for this installation from the choices below.

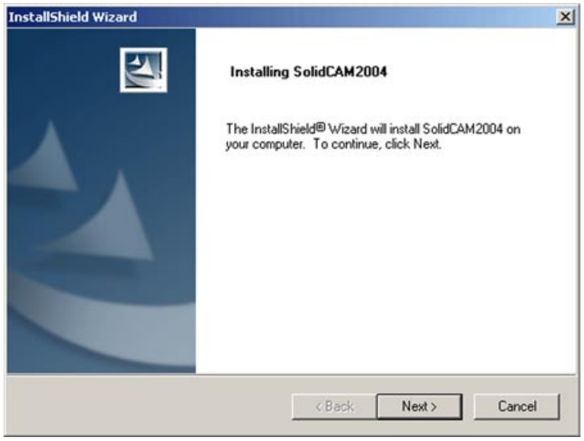
English

Choose **English** language.

When you have to install a localized version of SolidCAM, choose the appropriate language from the combo-box.

The installation procedure will be continued.

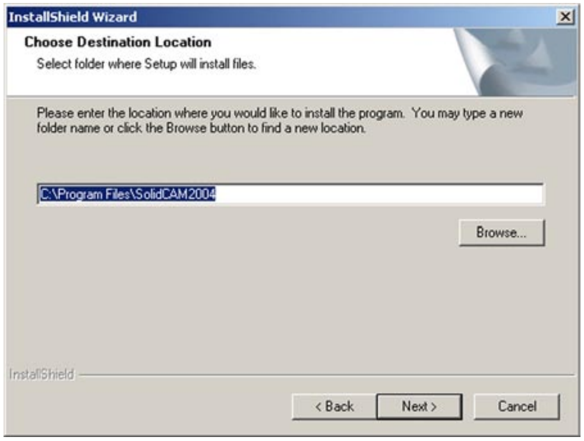
3. The **Installing SolidCAM** dialog will be displayed. Click on the **Next** button.


InstallShield Wizard
< Back
Next >
Cancel

Installing SolidCAM2004

The InstallShield® Wizard will install SolidCAM2004 on your computer. To continue, click Next.

4. The **Choose Destination Location dialog** will be displayed.


InstallShield Wizard
Browse...
InstallShield
< Back
Next >
Cancel

Choose Destination Location

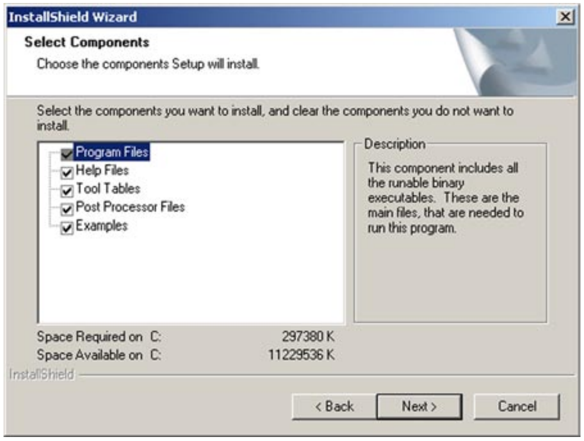
Select folder where Setup will install files.

Please enter the location where you would like to install the program. You may type a new folder name or click the Browse button to find a new location.

C:\Program Files\SolidCAM2004

This dialog enables you to choose the destination folder for SolidCAM. You have to either confirm the default destination folder (**C:\Program Files\SolidCAMXXXX**) with the **Next** button or choose other destination folder with the **Browse** button and confirm it with the **Next** button.

5. The **Select Components dialog** will be displayed.


InstallShield Wizard
Select Components
InstallShield
< Back
Next >
Cancel

Choose the components Setup will install.

Select the components you want to install, and clear the components you do not want to install.

☑ Program Files
☑ Help Files
☑ Tool Tables
☑ Post Processor Files
☑ Examples

Description

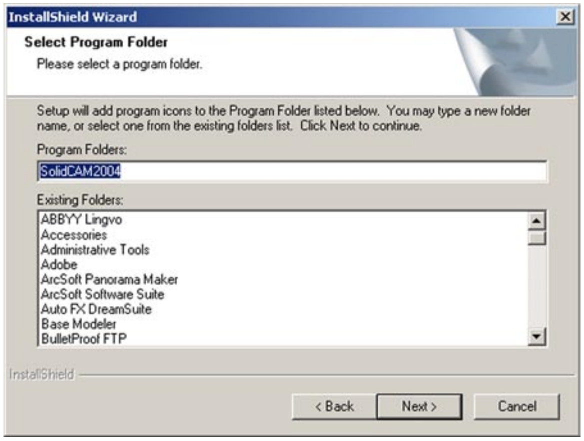
This component includes all the runable binary executables. These are the main files, that are needed to run this program.

Space Required on C: 297380 K
Space Available on C: 11229536 K

Choose the necessary SolidCAM component and confirm your choice with the **Next** button.

Make a note that **Program Files** component can not be unselected.

6. The **Select Program Folder** dialog will be displayed.


InstallShield Wizard
Select Program Folder
InstallShield
< Back
Next >
Cancel

Please select a program folder.

Setup will add program icons to the Program Folder listed below. You may type a new folder name, or select one from the existing folders list. Click Next to continue.

Program Folders:

SolidCAM2004

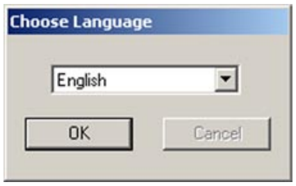
Existing Folders:

ABBYY Lingvo
Accessories
Administrative Tools
Adobe
ArcSoft Panorama Maker
ArcSoft Software Suite
Auto FX DreamSuite
Base Modeler
BulletProof FTP

Confirm the default **Program Folder** name - **SolidCAMXXXX** with the **Next** button.

SolidCAM installation will be continued.

7. When the installation will be finished, the **Choose language dialog** will be displayed.

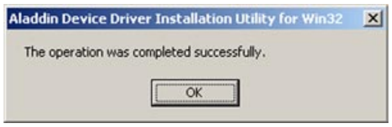


Choose Language
English
OK
Cancel

Choose **English** language and click on the **OK** button.

When you install a localized version of SolidCAM, choose the appropriate language from the combo-box.

8. SolidCAM installs drivers for hardware dongle. When the drivers will be installed, the following message will be displayed.



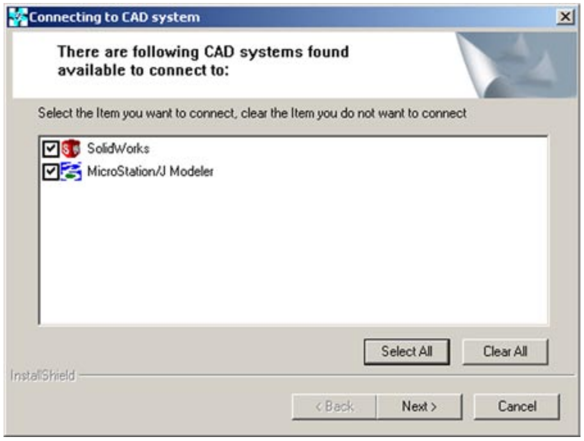
Aladdin Device Driver Installation Utility for Win32

The operation was completed successfully.

OK

Click on the **OK** button.

9. The **Connection to CAD system dialog** will be displayed. This dialog displays all CAD systems that can be used by SolidCAM.



Connecting to CAD system

There are following CAD systems found available to connect to:

Select the Item you want to connect, clear the Item you do not want to connect

☑ SolidWorks
☑ MicroStation/J Modeler

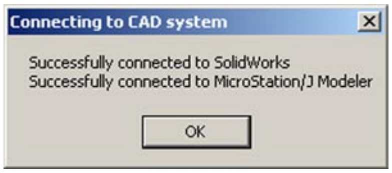
Select All
Clear All
InstallShield
< Back
Next >
Cancel

Choose the appropriate Host-CAD systems for SolidCAM.

When SolidCAM found only one CAD system available for connection, this dialog is not displayed.

Click on the **Next** button.

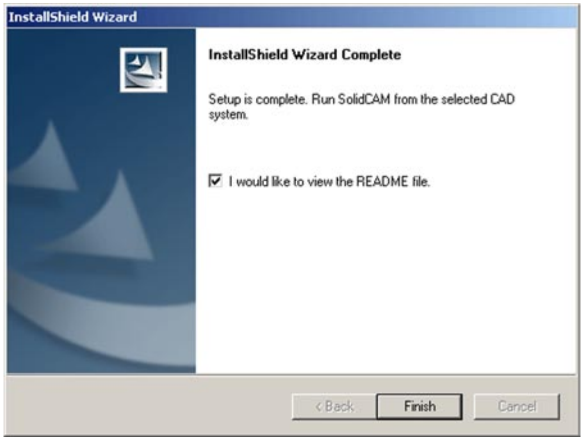
SolidCAM will be connected to the defined CAD systems; the following message will be displayed.


Connecting to CAD system
OK

Successfully connected to SolidWorks
Successfully connected to MicroStation/J Modeler

Click on the **OK** button.

10. The following dialog will be displayed.


InstallShield Wizard
< Back
Finish
Cancel

InstallShield Wizard Complete

Setup is complete. Run SolidCAM from the selected CAD system.

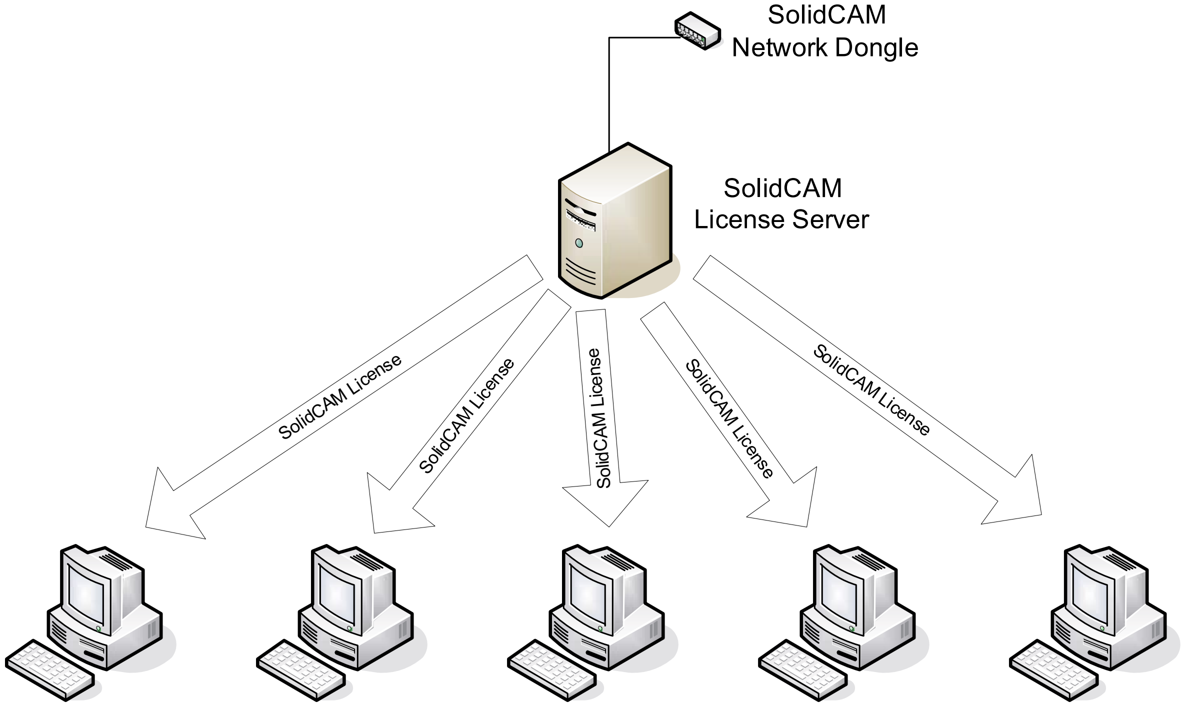
☑ I would like to view the README file.

Click on the **Finish** button.

The installation is finished.

1.1.3 SolidCAM Network License Installation

SolidCAM enables you to install number of SolidCAM copies through your network. The SolidCAM Network Dongle have to be connected the server. The License Manager will be installed on the network server computer, which will distribute SolidCAM Licenses through client computers.


SolidCAM
Network Dongle
SolidCAM
License Server
SolidCAM License
SolidCAM License
SolidCAM License
SolidCAM License
SolidCAM License

SolidCAM Client Computers

- Check with which protocol is used for your network. SolidCAM enables you to use TCP/IP protocol (recommended) or IPX protocol.
- Install SolidCAM as explained above on the computer that will be used as SolidCAM License Server.

TCP/IP Protocol

1. Open in a text editor the **NethaspTCPIP.ini** file located in the **..\SolidCAMXXXX\Util** folder.
2. In the **[NH_TCPIP]** section edit the value of the **NH_SERVER_ADDR** parameters.

NH_SERVER_ADDR = xx.xx.xx.xx;

3. Type the IP address of the network server instead of xx.xx.xx.xx.
4. Rename the **NethaspTCPIP.ini** file to **Nethasp.ini**.

IPX Protocol

Rename the **NethaspIPX.ini** file to **Nethasp.ini**.

Installation of SolidCAM License Server

1. Connect the NetHASP dongle to the parallel port of the computer that will be used as SolidCAM License Server.
2. Run the Installation utility of the HASP dongle driver. Choose the **Run** option from the windows **Start menu** and type the following command in the **Run dialog**: "**<path for your SolidCAM folder>\Util\hinstall.exe –i**".

   The HASP device driver will be installed. Restart the computer.
3. Install the NetHASP License Manager - **LMsetup.exe** located in the **..\SolidCAMXXXX\Util** folder. The NetHASP License Manager will be installed.
4. Activate the NetHASP License Manager. Choose the **NetHASP License Manager / NetHASP License Manager** command from the **Programs** menu.

If the NetHASP License Manager already installed on your server you can use it and don't need to install another copy of the NetHASP License Manager.

Installation of SolidCAM2004 on Client computers

1. Install SolidCAM on the client computer.
2. Copy the appropriate **NETHASP.INI** file from the server to the **..\SolidCAMXXXX\SolidCAM** folder.
3. Run SolidCAM.

1.1.4 SolidCAM Dongle Update

In some cases SolidCAM requires the update of SolidCAM dongles. To update the dongle do the following steps:

1. Make sure that the SolidCAM dongle is connected to the computer.
2. Extract the ZIP-archive **<dongle number>.zip** supplied by SolidCAM into the **..\SolidCAMXXXX\Util** folder
3. Extract the ZIP-archive **customer.zip** to the same folder.
4. Run the **Mak_plag.bat** file located in the **..\SolidCAMXXXX\Util** folder. It is recommended to run it with the **Command Prompt** (**Start > Programs > Accessories**) window. When the **Command prompt dialog** is displayed enter the following commands:

   ```
   Cd <full path for SolidCAM folder>\Util

   Mak_plag
   ```

   By default, the full path for SolidCAM folder is the following: **c:\Program Files\SolidCAMXXXX**
5. The message: **Hasp Memory updated OK** will be displayed.

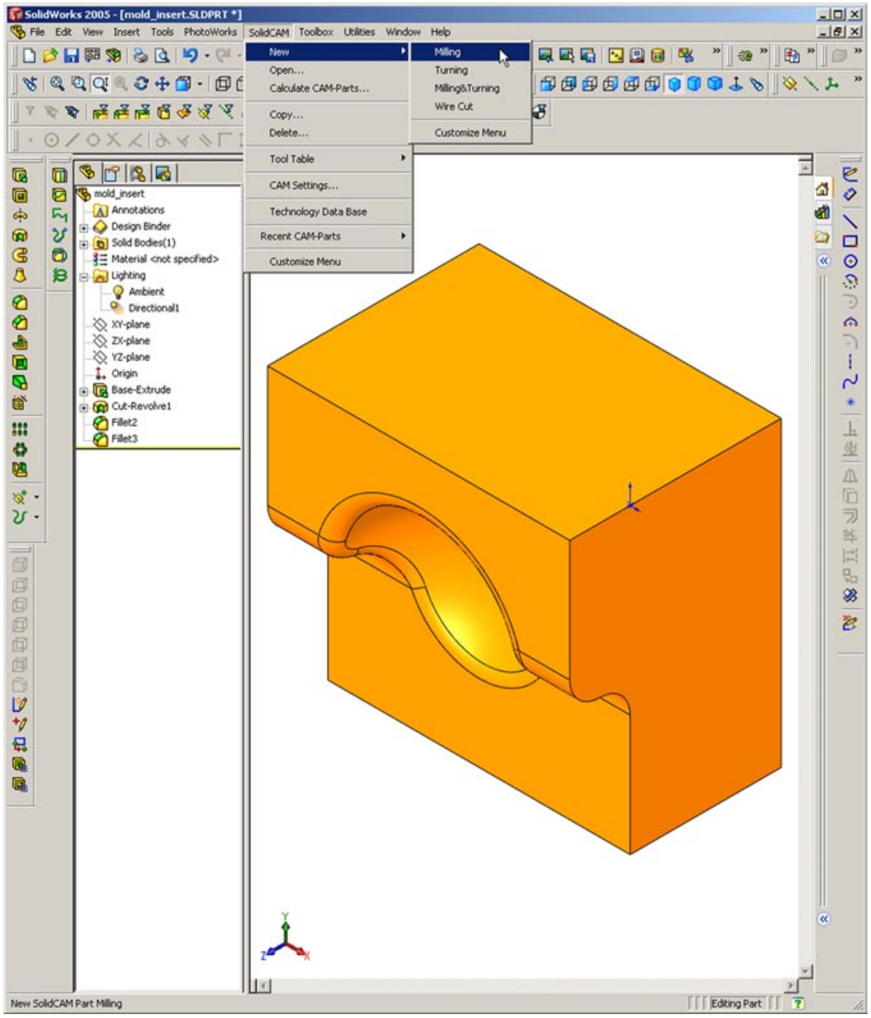
1.2 Basics Concepts

Every manufacturing project in SolidCAM contains the following data:

- **CAM-Part** - The CAM-Part defines the general data of the workpiece. This includes the model name, the home position, tool options, CNC-controller etc...
- **Geometry** - By selecting Edges, Curves, Surfaces or Solids, define **WHAT** and **WHERE** you are going to machine. This geometry is associated with the native SolidWorks model.
- **Job** - A Job is a single machining operation in SolidCAM. In the Job Technology, Tool parameters and Strategies are defined - in short **HOW** you want to machine.

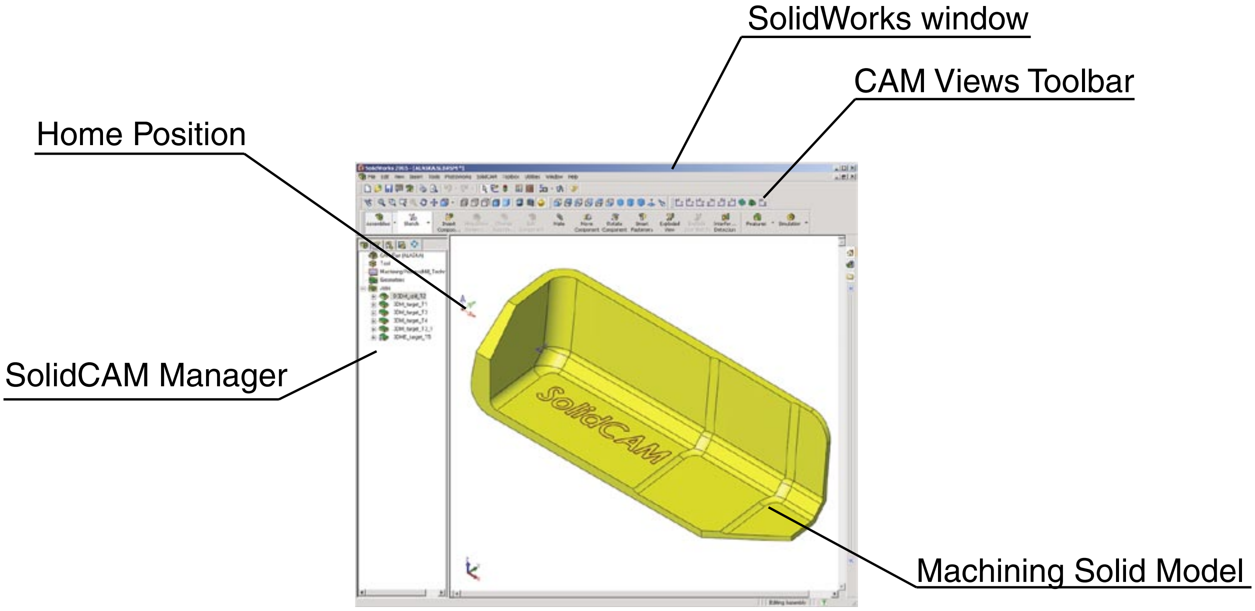
1.3 Starting SolidCAM

To activate SolidCAM, simply click on the field **SolidCAM** in the main menu of SolidWorks and choose **Milling** from the New sub-menu. SolidCAM will be started.

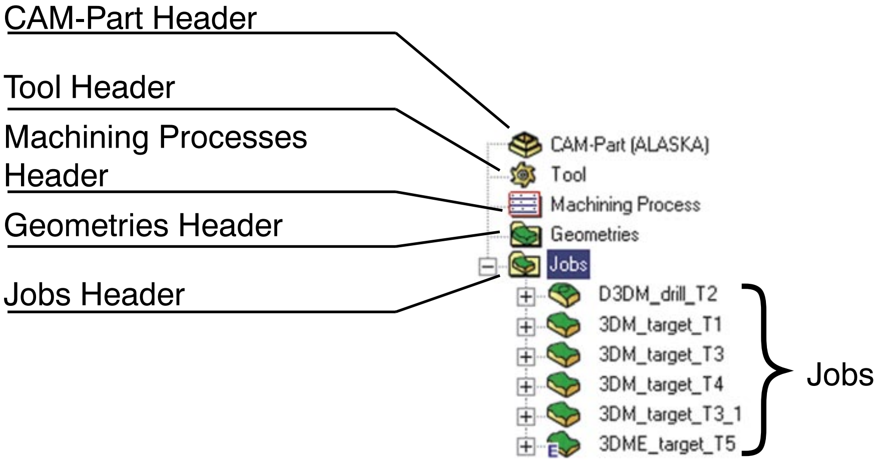

SolidWorks 2005 - [mold_insert.SLDPRT *]
File Edit View Insert Tools PhotoWorks SolidCAM Toolbox Utilities Window Help
New
Open...
Calculate CAM-Parts...
Copy...
Delete...
Tool Table
CAM Settings...
Technology Data Base
Recent CAM-Parts
Customize Menu
Milling
Turning
Milling&Turning
Wire Cut
Customize Menu
mold_insert
Annotations
Design Binder
Solid Bodies(1)
Material <not specified>
Lighting
Ambient
Directional1
XY-plane
ZX-plane
YZ-plane
Origin
Base-Extrude
Cut-Revolve1
Fillet2
Fillet3
New SolidCAM Part Milling
Editing Part

1.4 SolidCAM Interface

After the CAM-Part is loaded, the SolidCAM Manager will be displayed in the left part of the screen.


SolidWorks window
CAM Views Toolbar
Home Position
SolidCAM Manager
Machining Solid Model

The SolidCAM Manager is the main interface feature of SolidCAM. Complete information about the CAM-Part is shown in the SolidCAM Manager.


CAM-Part Header
Tool Header
Machining Processes Header
Geometries Header
Jobs Header
CAM-Part (ALASKA)
Tool
Machining Process
Geometries
Jobs
D3DM_drill_T2
3DM_target_T1
3DM_target_T3
3DM_target_T4
3DM_target_T3_1
3DME_target_T5
Jobs

The SolidCAM Manager contains the following fields:

- **CAM-part field**

  This field displays the name of the active **SolidCAM CAM-Part**. By right clicking on it, you can activate the CAM-Part menu to manage your CAM-Parts.

  For more information on CAM-Parts, please refer to **Chapter 2**.

- **Tool field**

  This field displays the name of the active **Tool Table**. By right clicking on it, you can activate the menu to manage the Tool Tables.

  For more information on Tools, please refer to **Chapter 3**.

- **Machining Process field**

  This field displays the name of the active **Machining Process Table**. By right clicking on it, you can activate the menu to manage the MP Tables.

  For more information on Machining Processes, please refer to the **Chapter 11**.

- **Geometries field**

  This field displays all the **SolidCAM geometries** not used in the Jobs. You can get the list of these geometries by clicking on the „**+**“ icon near the Geometries field. You can display the Geometries managing menu by right clicking on the Geometries field. You can also get the relevant menu by right clicking on each Geometry name.

  For more information on Geometries, please refer to **Chapter 4**.

- **Jobs field**

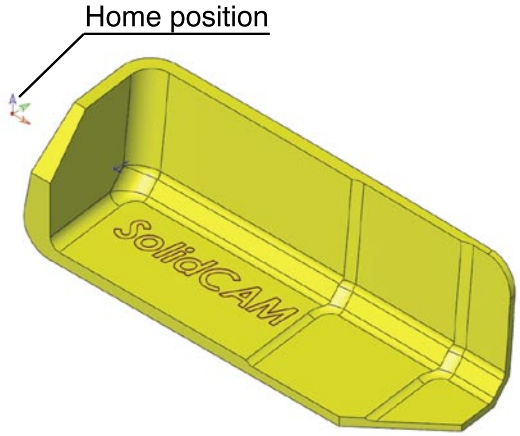
  This field shows you all the **SolidCAM Jobs**. The Jobs managing menu is available by right clicking on the Jobs field. You can get the relevant menu also by right clicking on each Job name.

  For more information on Jobs, please refer to **Chapter 5**.

Home position

The Home position is shown on the model. It defines the origin for all the machining operations of the CAM-Part.

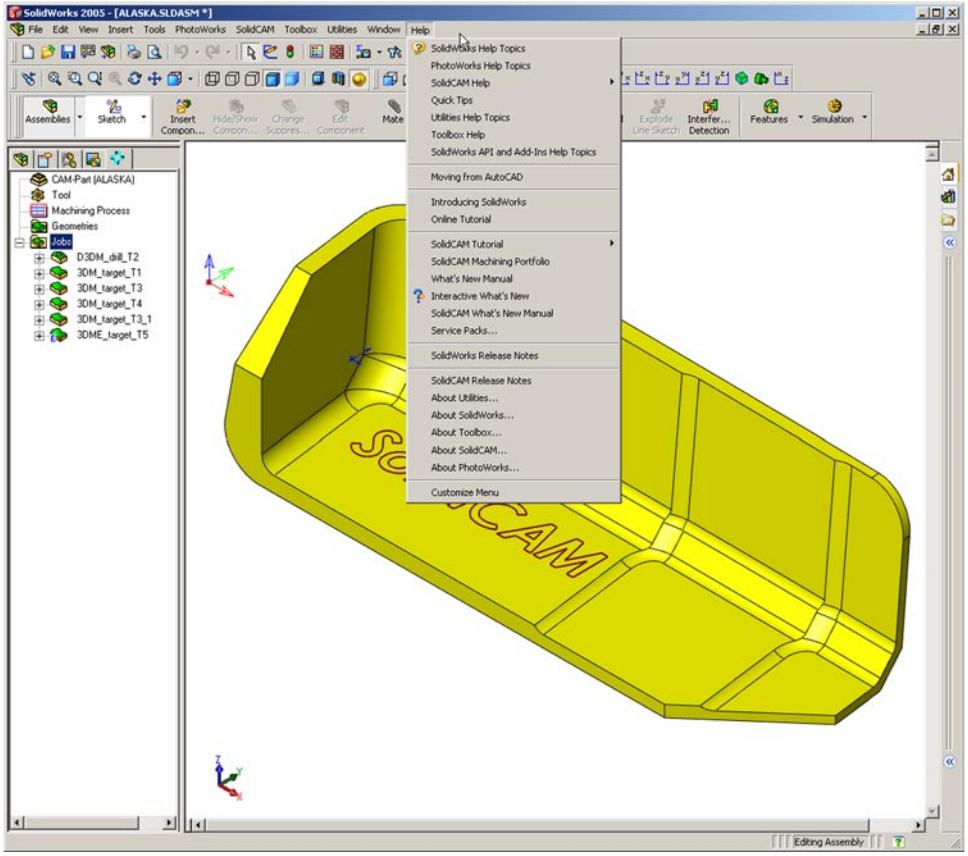
For more information on Homes, please refer to the topic **2.2**.



1.5 Getting Help

The SolidCAM On-Line Help enables you to get on-line help about different options in SolidCAM. This help is installed in the **.../Documentation** folder after the standard installation procedure.

This Help is available via the SolidWorks Help menu. Click on the **Help** item in the main menu of SolidWorks and choose the **SolidCAM Help -> Milling** items from the submenu.


SolidWorks 2005 - [ALASKA.SLDASM *]
File Edit View Insert Tools PhotoWorks SolidCAM Toolbox Utilities Window Help
SolidWorks Help Topics
PhotoWorks Help Topics
SolidCAM Help
Quick Tips
Utilities Help Topics
Toolbox Help
SolidWorks API and Add-Ins Help Topics
Moving from AutoCAD
Introducing SolidWorks
Online Tutorial
SolidCAM Tutorial
SolidCAM Machining Portfolio
What's New Manual
Interactive What's New
SolidCAM What's New Manual
Service Packs...
SolidWorks Release Notes
SolidCAM Release Notes
About Utilities...
About SolidWorks...
About Toolbox...
About SolidCAM...
About PhotoWorks...
Customize Menu
Assemblies
Sketch
Insert Compon...
Hide/Show Compon...
Change Suppres...
Edit Component
Mate
Explode Line Sketch
Interfer... Detection
Features
Simulation
CAM-Part (ALASKA)
Tool
Machining Process
Geometries
Jobs
D3DM_drill_T2
3DM_target_T1
3DM_target_T3
3DM_target_T4
3DM_target_T3_1
3DME_target_T5
Editing Assembly

SolidCAM Help is fully context sensitive. You can get specific Help topic for the each SolidCAM dialog by pressing the **F1** button. To get information about a specific dialog area like field, button etc... you have to click on the question-mark in the upper-right corner of the dialog and then click on the field or button you need.

The On-Line Help will be displayed and then the specific topic will be explained.

The Machining Portfolio is also available in the **.../Documentation** folder. This document provides an overview of the various features of SolidCAM.

CAM-Part

2

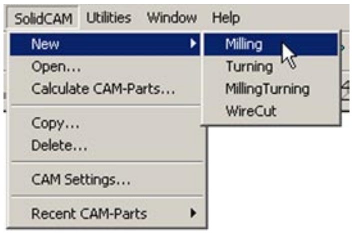
In the CAM-Part you have to specify all the information relevant to the machining project or workpiece you want to manufacture. The CAM-Part folder will include all machining geometries, job definitions and generated G-Code files.

The parameters and values defined at the CAM-Part level, such as Home position, Milling levels and Tools will serve as default values for the separate machining operations. All specified parameters can be changed in single operations (jobs).

- ❐ Starting a new Milling CAM-Part
- ❐ Managing CAM-Parts
- ❐ Defining a Home
- ❐ Defining Stock Model
- ❐ Defining Target Model
- ❐ CAM-Part Documentation

2.1 Starting a new Milling CAM-Part

When the SolidWorks Part is loaded SolidCAM Manager is loaded, do the following:

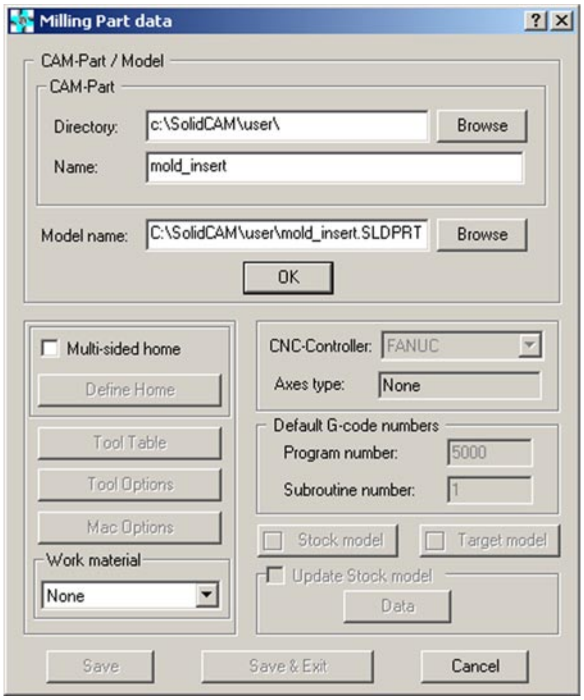

SolidCAM
Utilities
Window
Help
New
Open...
Calculate CAM-Parts...
Copy...
Delete...
CAM Settings...
Recent CAM-Parts
Milling
Turning
MillingTurning
WireCut

1. Click on the SolidCAM item in the SolidWorks menu. The SolidCAM main menu will be displayed.
2. Choose the **New** option from the menu to define a new CAM-Part.
3. Choose the type of new CAM-Part - **Milling** from the submenu.
4. The **CAM-Part data dialog** will be displayed.

When you start to program a CAM-Part, you have to decide on:

- What type of CNC machine you are going to use? (3, 4 or 5 axis).
- Where are you going to clamp this part?
- Where are your main home positions? (On a 4 or 5 axis CNC machine every clamping position is the main home to which all additional positioning are related).

In the CAM-Part you have to specify all the information relevant to the machining project or workpiece you want to manufacture. The parameters and values defined at the CAM-Part level will serve as default values for machining levels, etc. Naturally, all parameters can be changed in single machining operations (jobs).



Directory: c:\SolidCAM\user\
Name: mold_insert
Model name: C:\SolidCAM\user\mold_insert.SLDPRT
☐ Multi-sided home
CNC-Controller: FANUC
Axes type: None
Program number: 5000
Subroutine number: 1
☐ Stock model
☐ Target model
☐ Update Stock model
Work material: None

When you create a new CAM-Part, you have to enter a name for the CAM-Part and the model that contains the CAM-Parts geometry.

- **Directory**

  Specify the location of the CAM-Part. The default directory is the SolidCAM user directory (defined in the **SolidCAM Settings**). You can enter the path or use the **Browse** button to define the location.

- **CAM-Part name**

  Enter the name of the CAM-Part. You can give any name to identify your machining project. By default SolidCAM uses the name of the design model.

- **Model name**

  This field shows the name and location of the SolidWorks design model that you are using for the CAM-Part definition. The name is, by default, the name of the active SolidWorks document. With the **Browse** button you can choose any other SolidWorks document to define the CAM-Part. In this case the chosen SolidWorks document will be loaded into SolidWorks.

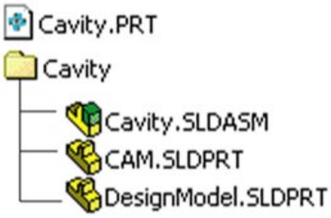
Every time the CAM-Part is opened, SolidCAM automatically checks the correspondence of the dates of the CAM-Part and the original SolidWorks design model. When the date of the original SolidWorks model is later than the date of the CAM-Part creation, this means that the SolidWorks original model has been updated. You can then replace the SolidWorks design model on which the CAM-Part is based with the updated SolidWorks design model.

After the **Directory**, **CAM-Part name** and **Model name** are defined, click on the **OK** button to confirm the CAM-Part creation. The CAM-Part will be defined and it‘s **structure** will be created.

2.1.1 The structure of the CAM-Part

The CAM-Part includes the following data (e.g. for a CAM-Part whose name is **Cavity**):

- A file **Cavity.prt** is located in the **SolidCAM User directory**.
- A subdirectory **Cavity** containing all the data generated for the CAM-Part.



Cavity.PRT
Cavity
  Cavity.SLDASM
  CAM.SLDPRT
  DesignModel.SLDPRT

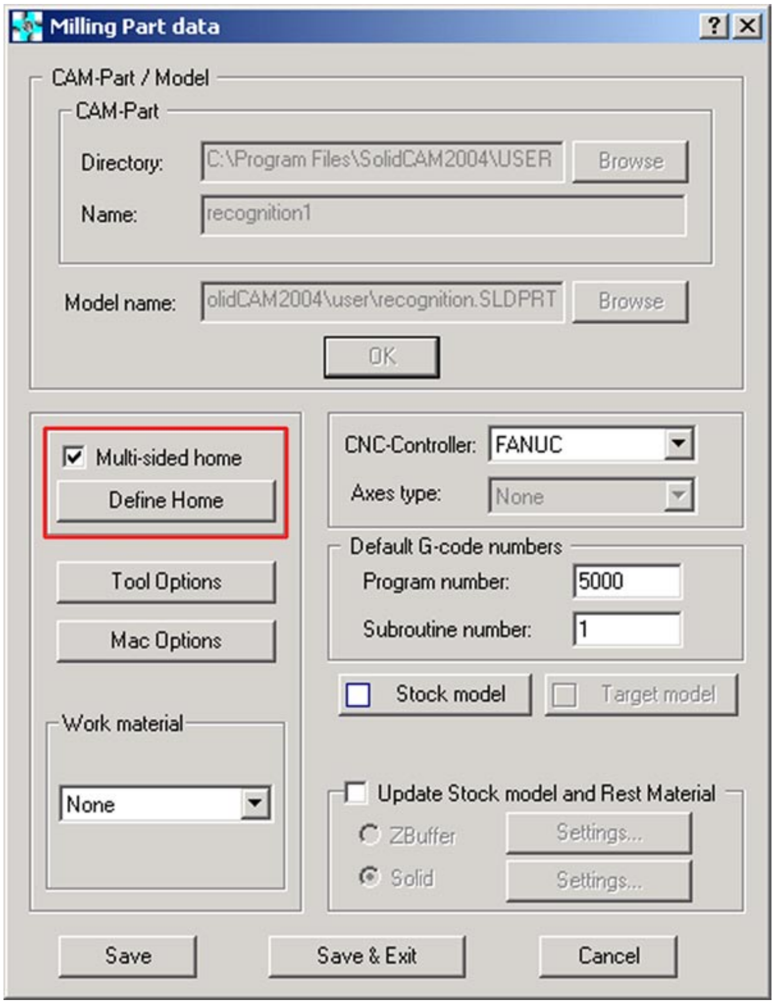
SolidCAM copies the original SolidWorks model to the subdirectory **Cavity** and creates a SolidWorks assembly that has the same name as the CAM-part (**Cavity.sldasm**). There are two components in this assembly:

DesignModel.sldprt - copy of the SolidWorks model file.

CAM.sldprt - contains SolidCAM home and geometry data.

SolidCAM CAM-Part uses the assembly environment of SolidWorks. This enables you to create auxiliary geometries (e.g. sketches) without making changes in the original design model. You can also insert some additional components into the assembly file such as stock model, CNC machine table, clamping and other tooling elements.

2.2 Defining Home


Milling Part data
CAM-Part / Model
CAM-Part
Browse
Browse
OK
Define Home
Tool Options
Mac Options
Work material
Default G-code numbers
Settings...
Settings...
Save
Save & Exit
Cancel

Directory: C:\Program Files\SolidCAM2004\USER

Name: recognition1

Model name: olidCAM2004\user\recognition.SLDPRT

☑ Multi-sided home

Work material: None

CNC-Controller: FANUC

Axes type: None

Program number: 5000

Subroutine number: 1

☐ Stock model

☐ Target model

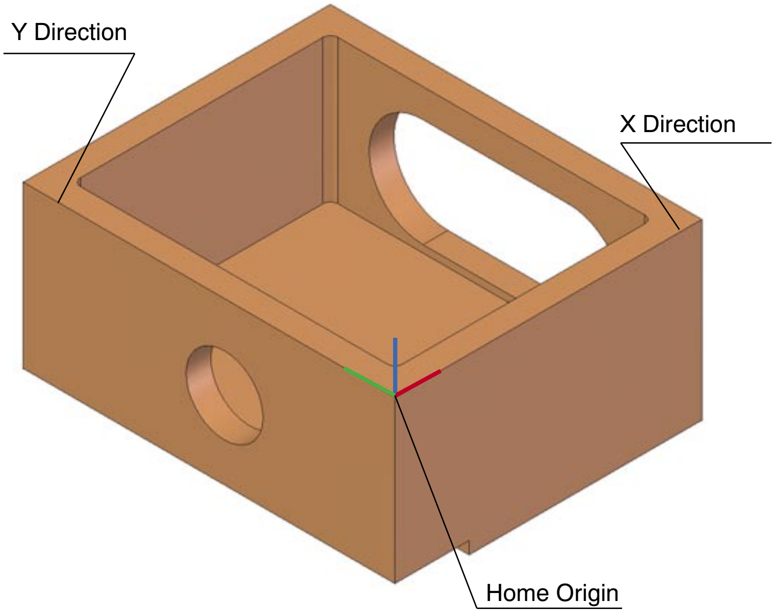
☐ Update Stock model and Rest Material

○ ZBuffer

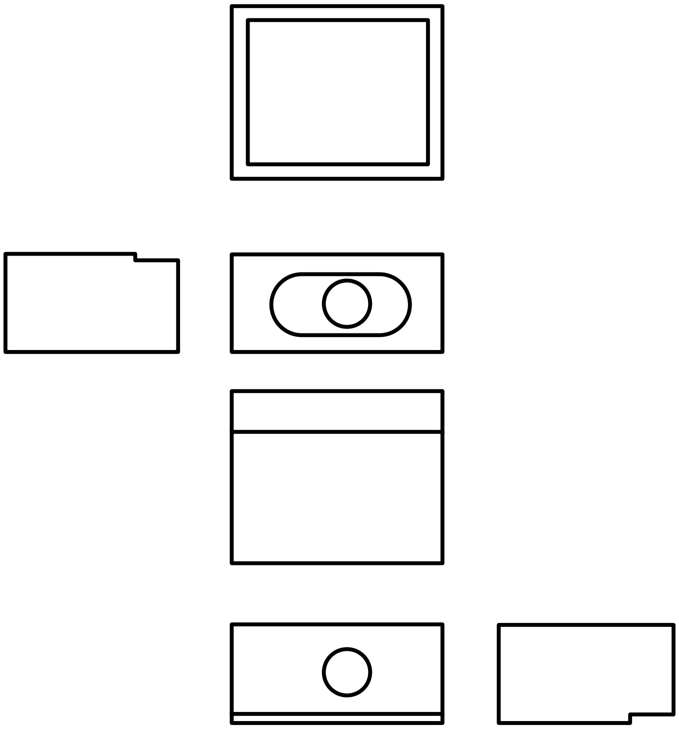
● Solid

2.2.1 Multi-Sided Home

This mode enables you to create CAM-Part defined directly on the solid model. You can define the home origin position and axes orientation by choosing points on the solid model or by selecting an already defined SolidWorks coordinate system.


Y Direction
X Direction
Home Origin

If this option is not chosen (**Projections mode**) SolidCAM enables you to create the CAM-Part on planar views defined by 2D sketches on the XY plane. In most cases this mode will be used when importing a DXF or IGES file and when you do not want to create a solid model.

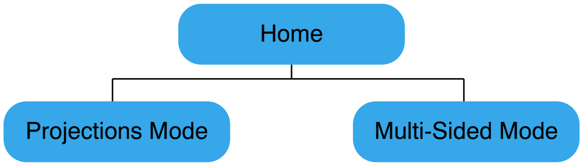


You can use this mode to create 3-, 4-, or 5-axis CAM-parts. After defining the main home on the XY-plane of SolidWorks, you must use the **Home Data dialog** to manually define the shifts and rotations of the additional positionings, from the main home.

2.2.2 Define Home

The Home position defines the origin for all machining operations of the CAM-Part. You can create multiple home positions and in each machining step select which home you want to use for the job. To complete the CAM-Part menu, you **must** enter the home position.

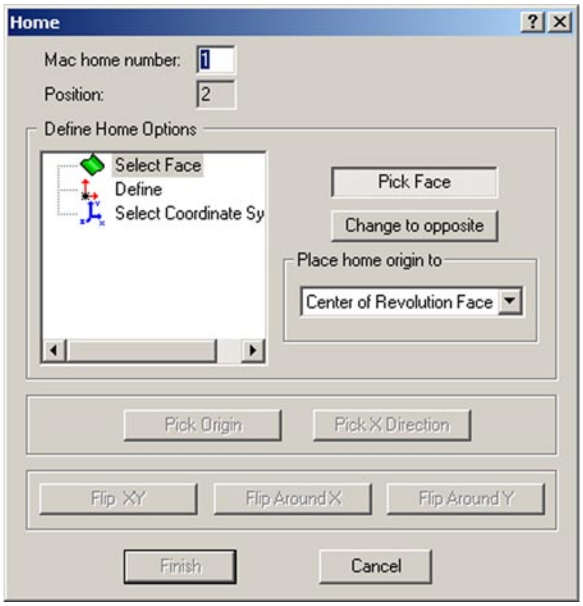
Click on the **Home** button in the **CAM-Part Data dialog**. If you define the first home in the current CAM-Part, the Home dialog will be displayed depending on your **Multi-Sided Home** settings.



If the Home that you define is not the first, the **Home Edit dialog** (see the topic **2.3.**) will be displayed. This dialog enables you to manage your Homes.

2.2.3 Add Home within Multi-Sided Mode

This mode enables you to define a **new** 3-4-5 axis home directly on the solid model. The newly created home will automatically receive the next sequential number.



Home

Mac home number: 1

Position: 2

Place home origin to: Center of Revolution Face

Define Home Options
Select Face
Define
Select Coordinate Sy
Pick Face
Change to opposite
Pick Origin
Pick X Direction
Flip XY
Flip Around X
Flip Around Y
Finish
Cancel

- **Mac home number** – this number corresponds with the built-in controller Home functions (example: G54, G55 etc… in the Fanuc type controllers). It can be used for different clamping positions (main homes) in different operations on the CAM-Part.

- **Position** – this number defines the sequence number of the home. For each MAC Home (main home) , several **Positions** are defined for different positionings; each such **Position** is related to the main Home.

- In multi-sided machining on 4- and 5-axis CNC machines, the SolidCAM postprocessor uses the specific controller Macro language to create the positioning relative to the main home.

- When you change the Mac home number to a new number in a 4-axis CNC machine, you have to define 3 points: Origin /X direction/ Y direction. If there are two or more CAM-Part home numbers defined on this Mac home, you will have to define only the Origin and a point on the plane to rotate to; the rotation axis is defined by the type of the 4-axis (_4th_axes_around = X / Y in the Mac file).

SolidCAM enables you to define the Home by one of the three methods:

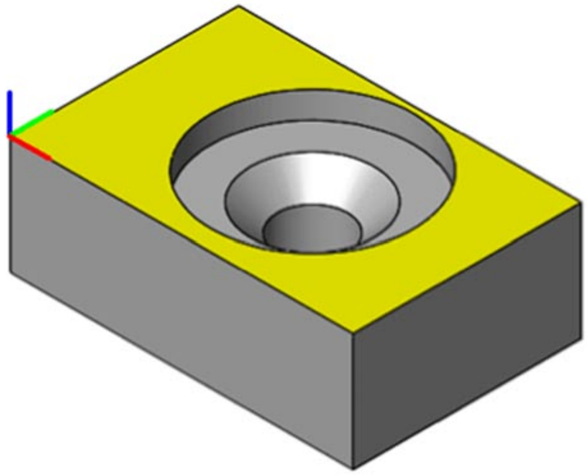
- **Select Face**
- **Define**
- **Coordinate system**

Select Face

SolidCAM enables you to define a new Home by selecting a face. The face can be one of the following:

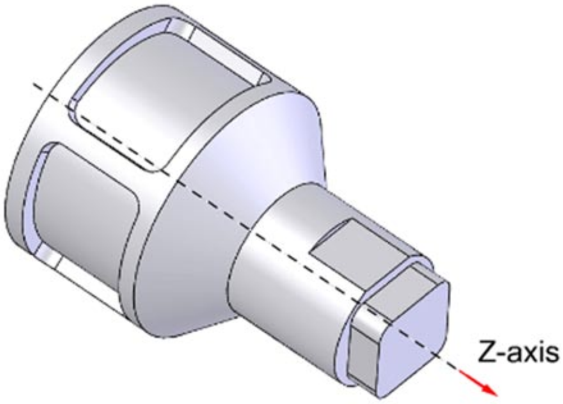
- **Planar face**

  In this case the Z-axis of the Home will be normal to the face.



- **Cylindrical/Conical face**

  In this case the Z-axis of the Home will be parallel to the axis of the revolution of the specified cylindrical/conical face.



Pick Face

This button enables you to select the face on the model for the Home definition. The selected model face will be higlighted.

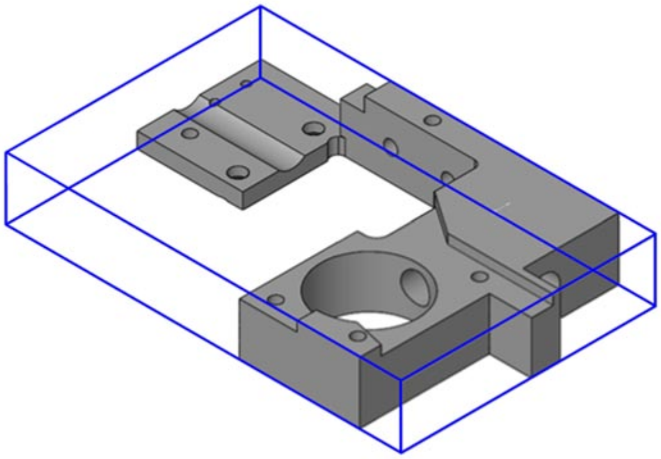
Change to opposite/Change to original

This button enables you to change the direction of the Z-axis of the home to the opposite direction.

Place home origin to:

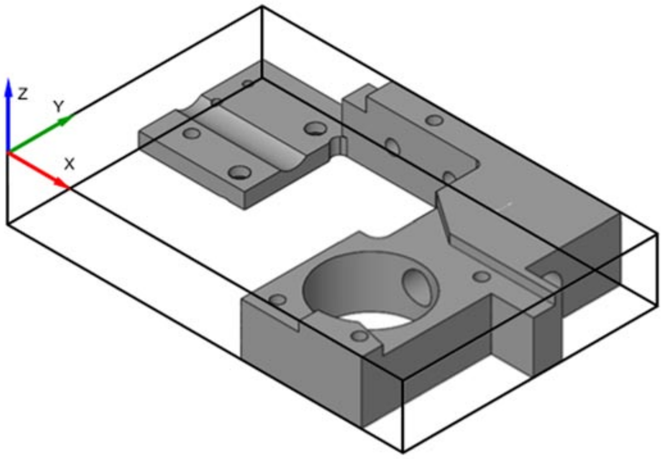
- **Corner of Model Box**

In this case the box surrounding the model will be calculated.



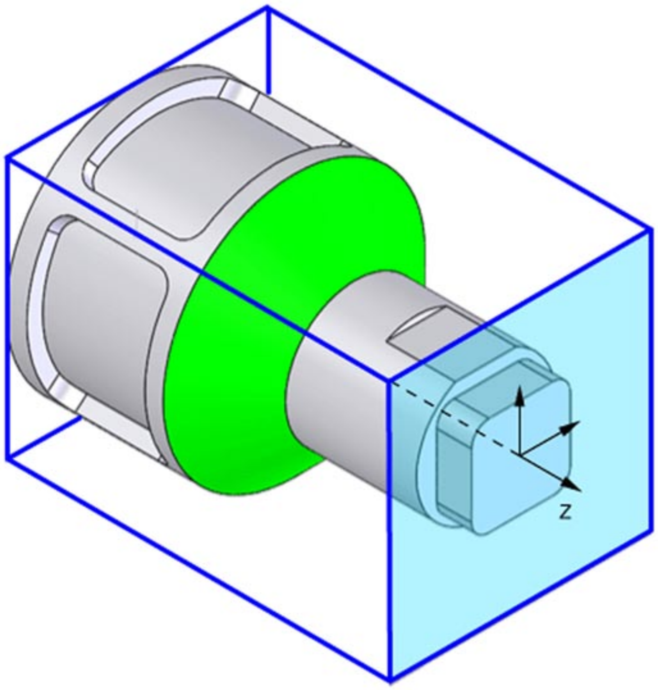
The upper plane of the model box is parallel to the XY-plane of the defined Home.

The Home Position will be located in the corner of the model box with the following coordinates (X_{MIN}, Y_{MIN}, Z_{MAX}).



- **Center of Revolution Face**

The origin will be located at the intersection of the rotation axis of the part and the face of the model box with a maximal Z coordinate (the Z-axis is directed along the rotation axis).

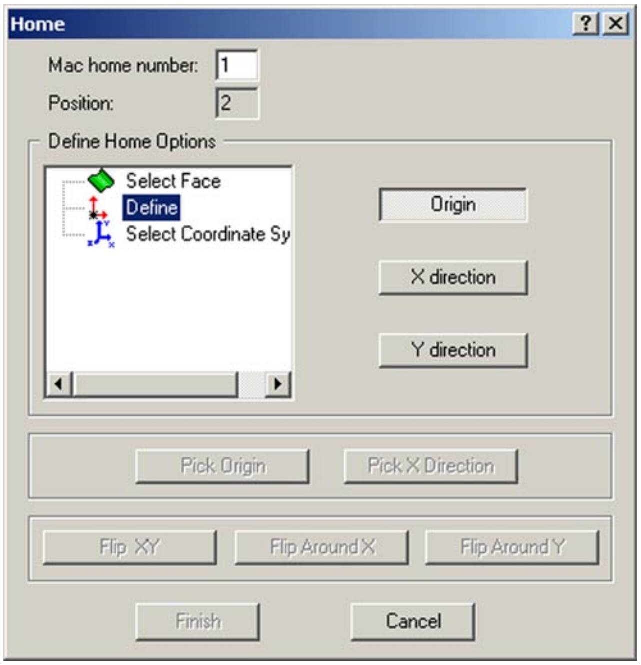


- **Home #1**

The Home origin will be in the same location of the **Home #1** origin.

This option is active only if the first Home is already defined.

Define


Home
Mac home number: 1
Position: 2
Define Home Options
Select Face
Define
Select Coordinate Sy
Origin
X direction
Y direction
Pick Origin
Pick X Direction
Flip XY
Flip Around X
Flip Around Y
Finish
Cancel

Origin – Select the origin point on the graphic screen.

X direction – Select a point relative to the origin that defines the X-axis.

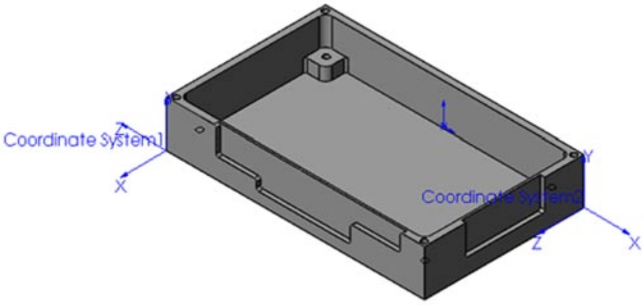
Y direction – Select a point (3rd point) that defines the plane. (The Y-axis will be 90 degrees to the X-axis; the selected point defines the plane).

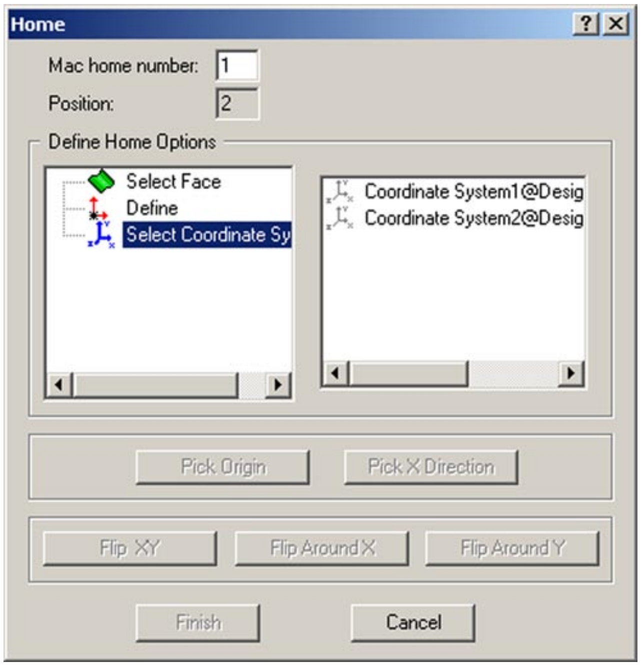
Pick Origin - This option enables you to define a new location for the Home Origin. The axis direction will not be changed.

Pick X direction - This option enables you to choose a new direction for the X-axis.

After a point is selected, the next button is automatically activated. If you miss the selection, you can, at any time, select the button you want to define and continue automatically to the next button.

Select coordinate system


Coordinate System1
X
Z
Coordinate System2
Y
Z
X


Home
Define Home Options
Select Face
Define
Select Coordinate Sy
Coordinate System1@Desig
Coordinate System2@Desig
Pick Origin
Pick X Direction
Flip XY
Flip Around X
Flip Around Y
Finish
Cancel

Mac home number: 1

Position: 2

SolidCAM enables you to choose the coordinate system defined in the CAD model file as the home.

Flip

SolidCAM enables you to flip the defined Home.

- **Flip XY**

  Pushing this button will rotate the Z-axis 180 degrees by replacing the X and Y between themselves (X becomes Y and Y becomes X).

- **Flip X**

  This button rotates the Home 180 degrees around the X axis.

- **Flip Y**

  This button rotates the Home 180 degrees around the Y axis.

Finish

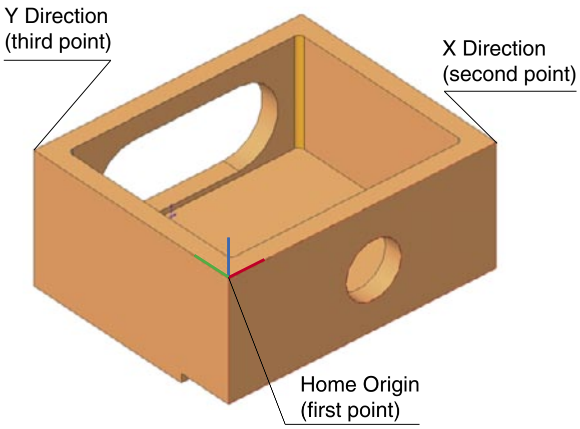
Pushing this button will add the home to the CAM-Part; the **Home data dialog** will be displayed.

2.2.4 Defining the homes for 3-axis CNC machine within Multi-Sided mode

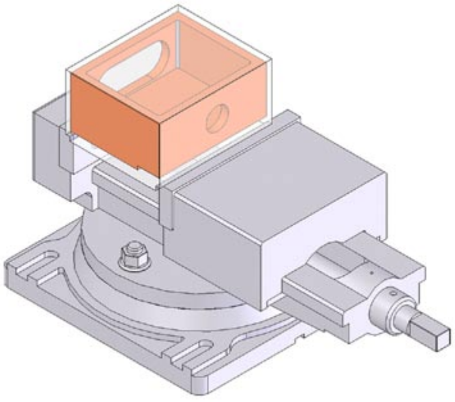
During the Home definition for 3-axis CNC machine you have to define three points with the **Home dialog**.

- **Origin** – Select the origin point on the graphic screen.
- **X direction** – Select a point relative to the origin that defines the X-axis.
- **Y direction** – Select a point (3rd point) to finish the definition of the XY-plane.

Use the mouse to define the origin, X and Y direction. You don't have to click on the **X direction** or the **Y direction** buttons in the **Home dialog**; they will be automatically selected after you define a point.

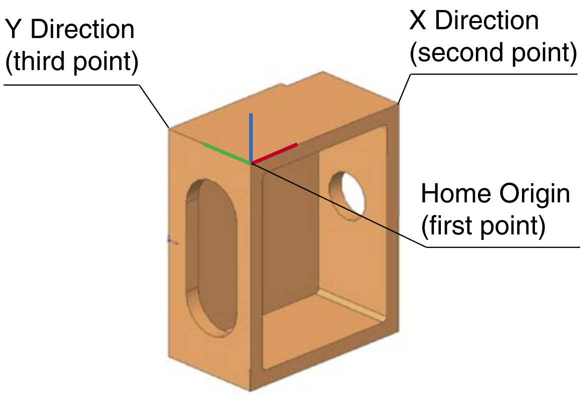


Home definition

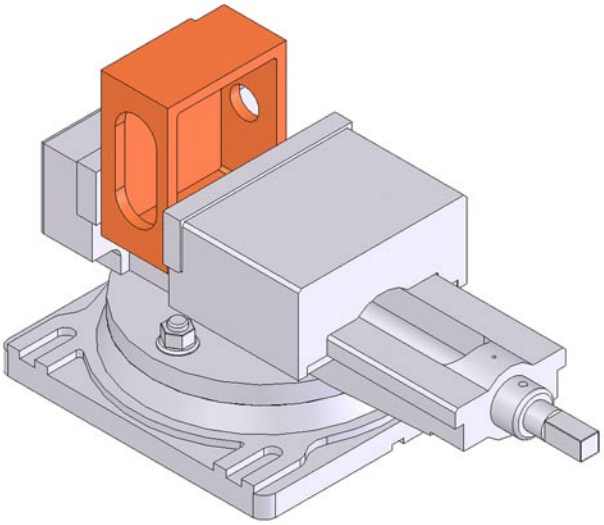


Clamping

For the next home position you also have to define three points.



Home definition



Clamping

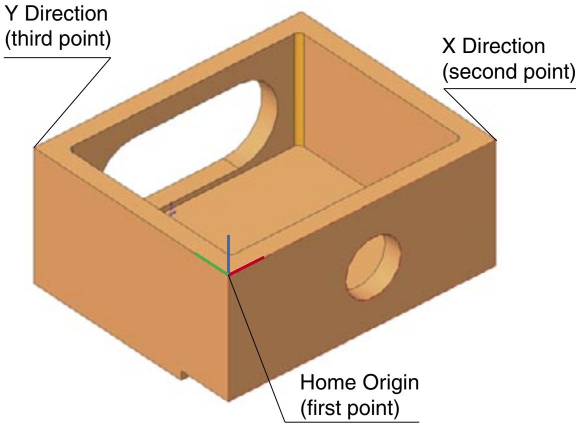
SolidCAM automatically assigns the new **Part Home number** for the each Home; the **Mac home number** can be changed by user for a new clamping (main home position).

2.2.5 Defining the homes for 4-axis CNC machine within Multi-Sided mode

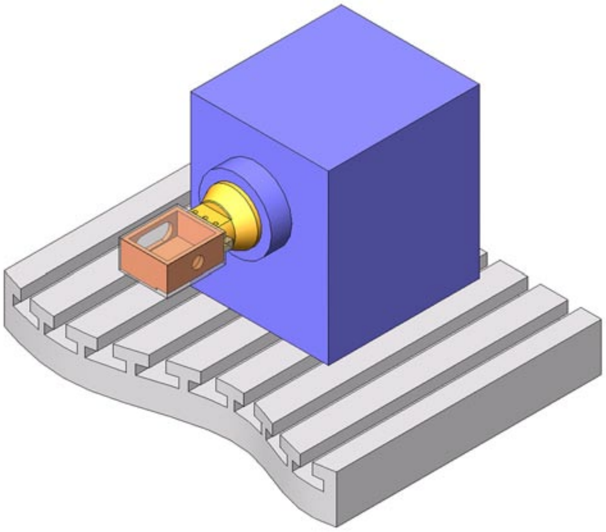
When you define the main home position (**Mac Home**) for the 4-axis CNC-machine you have to define three points with **Home dialog**:

- **Origin** – Select the origin point on the graphic screen.
- **X direction** – Select a point relative to the origin that defines the X-axis.
- **Y direction** – Select a point (3rd point) to finish the definition of the XY-plane.

Use the mouse to define the origin, X and Y direction. You don't have to click on the **X direction** or the **Y direction** buttons in the **Home dialog**; they will be automatically selected after you define a point.


Y Direction
(third point)
X Direction
(second point)
Home Origin
(first point)

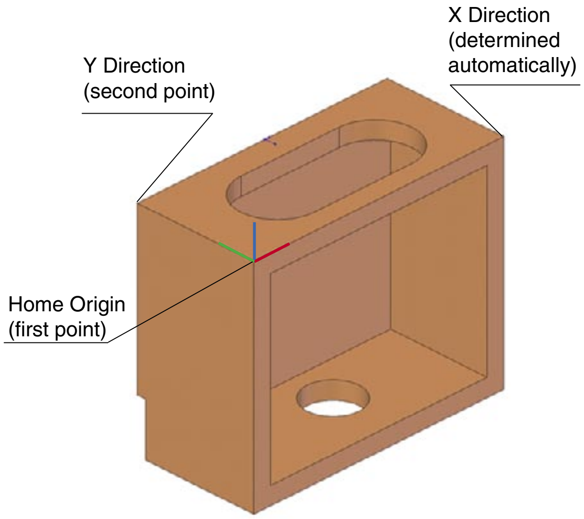
Home definition



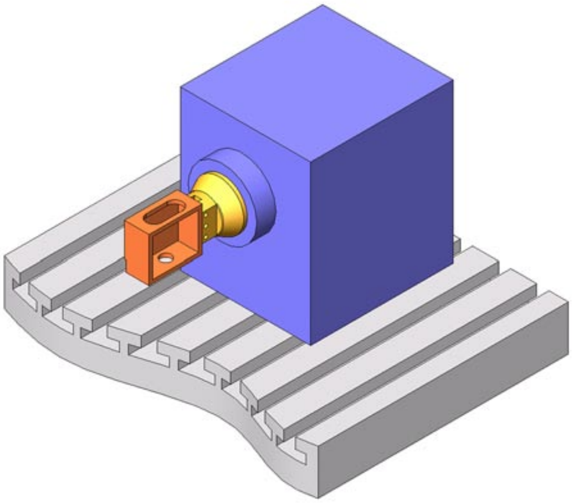
Clamping

To add a new **Part Home** you have to define only two points:

- The point for the **Home** origin.
- Point on the plane to rotate to. If the rotation axis is **X**, you have to define the **Y direction**. If the rotation axis is **Y**, you have to define the **X direction**.


X Direction
(determined
automatically)
Y Direction
(second point)
Home Origin
(first point)

Home definition



Clamping

SolidCAM automatically assigns the new **Part Home number** for the each Home; the **Mac home number** can be changed by user for a new clamping (main home position).

The rotation axis is defined automatically by SolidCAM according to the **Axis type (**see the topic **2.8)**.

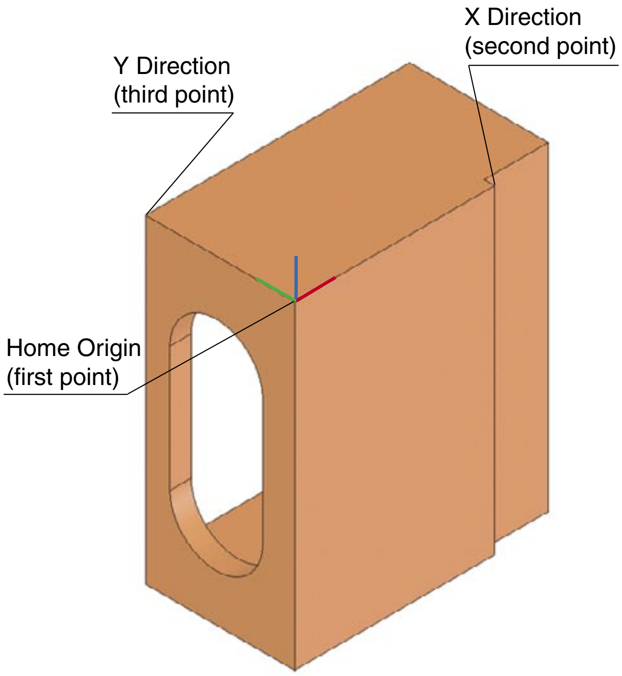
The **Shift** and **Rotation around** parameters will be calculated automatically.

2.2.6 Defining the homes for 5-axis CNC machine within Multi-Sided mode

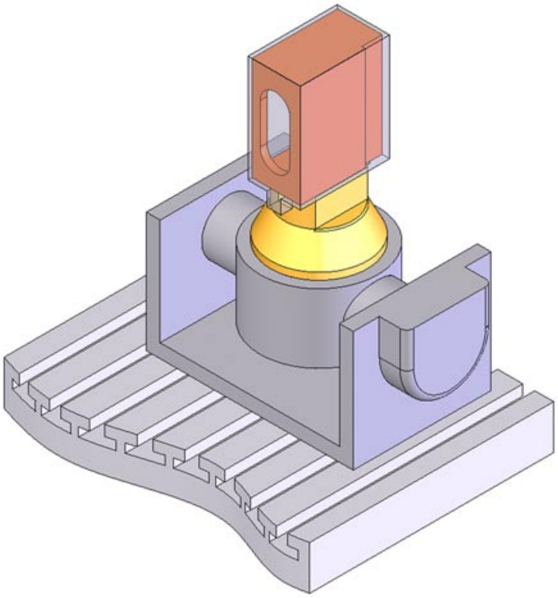
When you define the main home position (**Mac Home**) for the 4-axis CNC-machine you have to define three points with **Home dialog**:

- **Origin** - Select the origin point on the graphic screen.
- **X direction** - Select a point relative to the origin that defines the X-axis.
- **Y direction** - Select a point (3rd point) to finish the definition of the XY-plane.

Use the mouse to define the origin, X and Y direction. You don‘t have to click on the **X direction** or the **Y direction** buttons in the **Home dialog**; they will be automatically selected after you define a point.

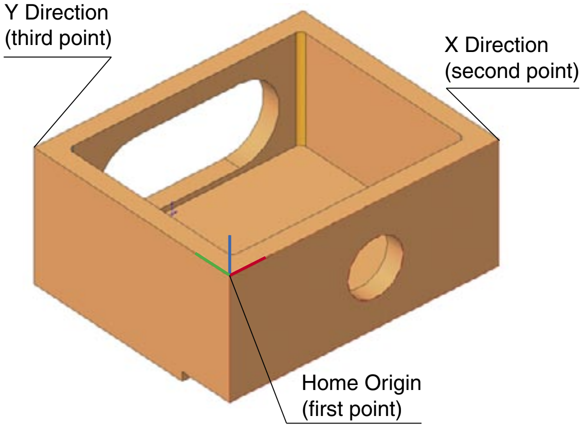


Home definition

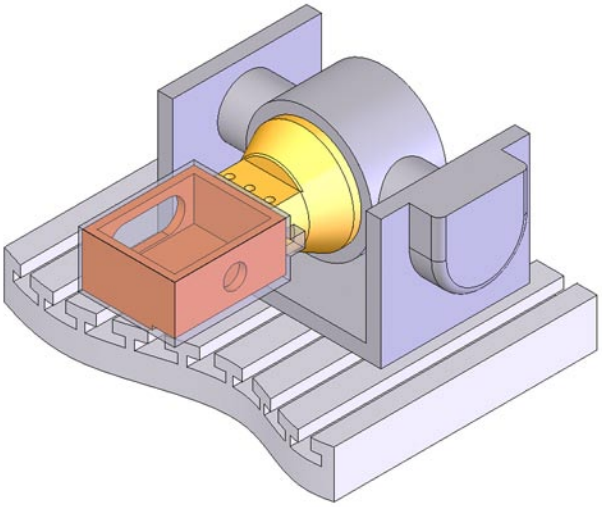


Clamping

To add a new **Part Home** you have to define three points.


Y Direction
(third point)
X Direction
(second point)
Home Origin
(first point)

Home definition



Clamping

SolidCAM automatically assigns the new **Part Home number** for the each Home; the **Mac home number** can be changed by user for a new clamping (main home position).

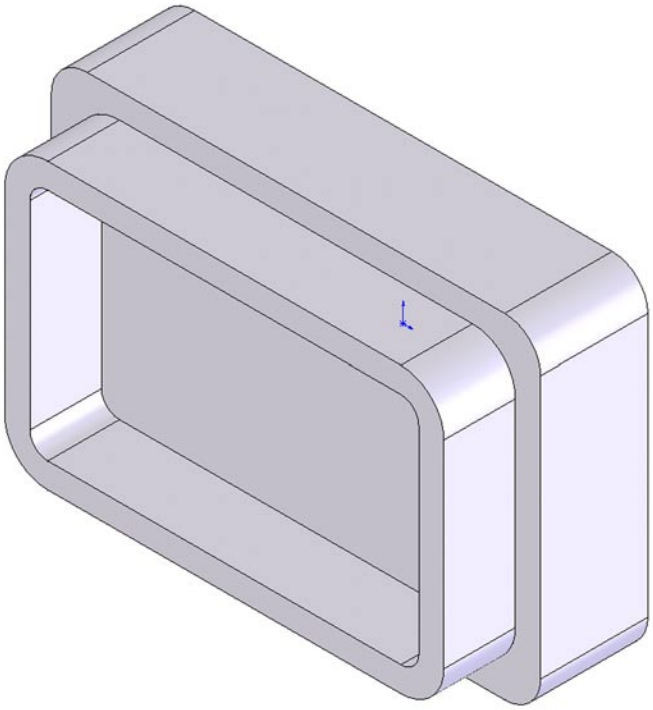
The rotation axis is defined automatically by SolidCAM according to the **Axis type**. (see the topic **2.8**)

The **Shift** and **Rotation** around parameters will be calculated automatically.

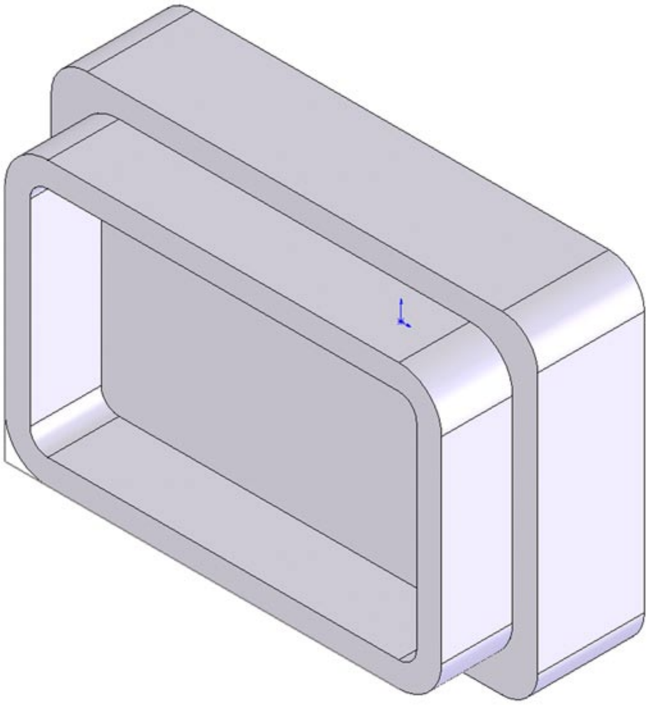
2.2.7 Home definition methods

Home definition using a sketch

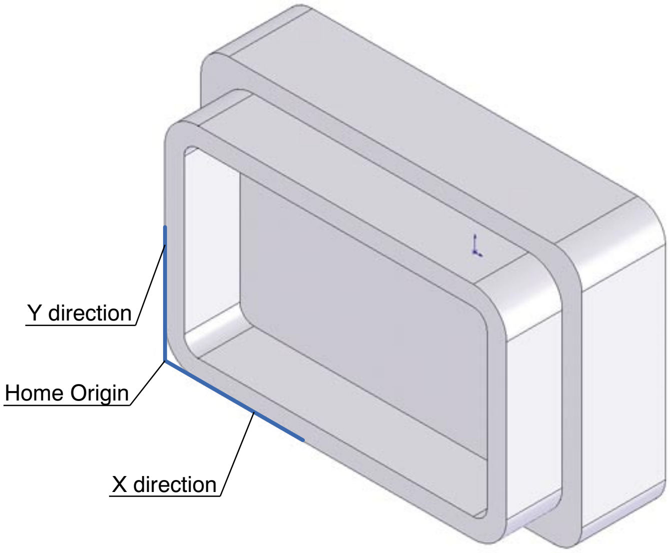
In some cases you cannot define the Home origin position or the direction of the axes without adding a geometry.



In this case you have to create a sketch that contains two perpendicular lines; the intersection point of the lines will define the Home origin and the lines define the X- and Y-axis directions.

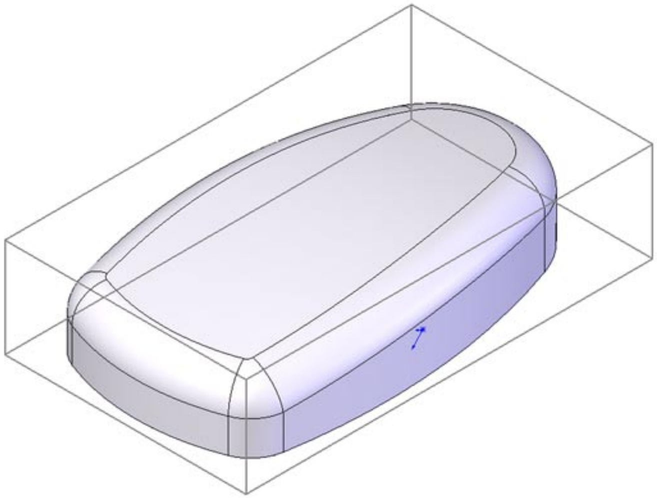


Choose the origin location at the intersection point of the lines and define the axis direction.

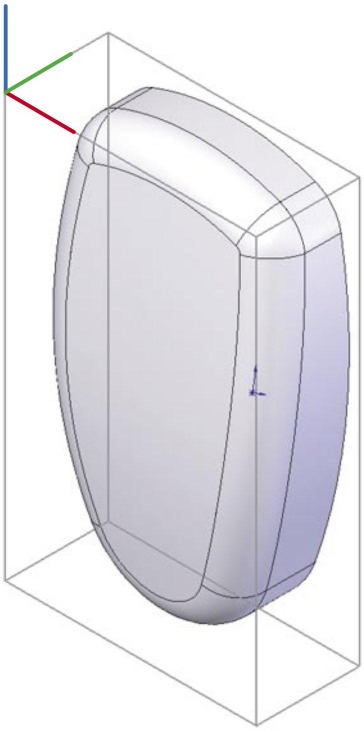


Home definition using a stock box

SolidCAM enables you to automatically generate the **stock box surrounding the model** with a defined offset. (see the topic **2.4.2**) The top stock plane will be parallel to the XY plane of the machine coordinate system.



Define the **Home** in the stock box corner.



Sometimes the model is located in the assembly in such a way that you need to rotate it before using the **Stock by box** option in order to enable SolidCAM to calculate the optimal stock.

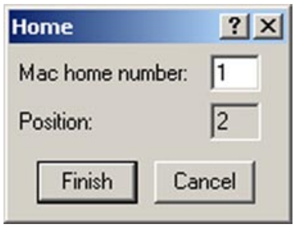
2.2.8 Add Home within Projections Mode

We will use this method for imported DXF or 2D IGES files. In this exercise the way to work with this mode on 2D sketches of the working views will be explained.

You can use this method for solids only if you are going to mill only from 1 direction and the **orientation** of the solid is exactly like in the milling position.

In the **Projections mode** you have to use a lot of imagination and knowledge in the multi-axial positioning in order to define the correct coordinate for the Home positions.

1. Define **Mac home number** – the machine home number that has to be set in the CNC controller (Home #1 = G54).

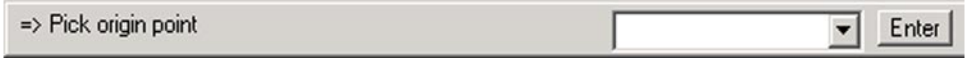


Mac home number: 1

Position: 2

- **Mac home number** – this number corresponds with the built-in controller Home functions (example: G54, G55 etc… in the Fanuc type controllers). It can be used for different clamping positions (main homes) in different operations on the CAM-Part.
- **Position** – this number defines the sequence number of the home. For each MAC Home (main home), several **Positions** are defined for different positionings; each such **Position** is related to the main Home.

2. **Pick home position** - Pick a point - or enter the X, Y, Z coordinate values of the CAM-Part's home position into the **Edit bar** and confirm by clicking on Enter.

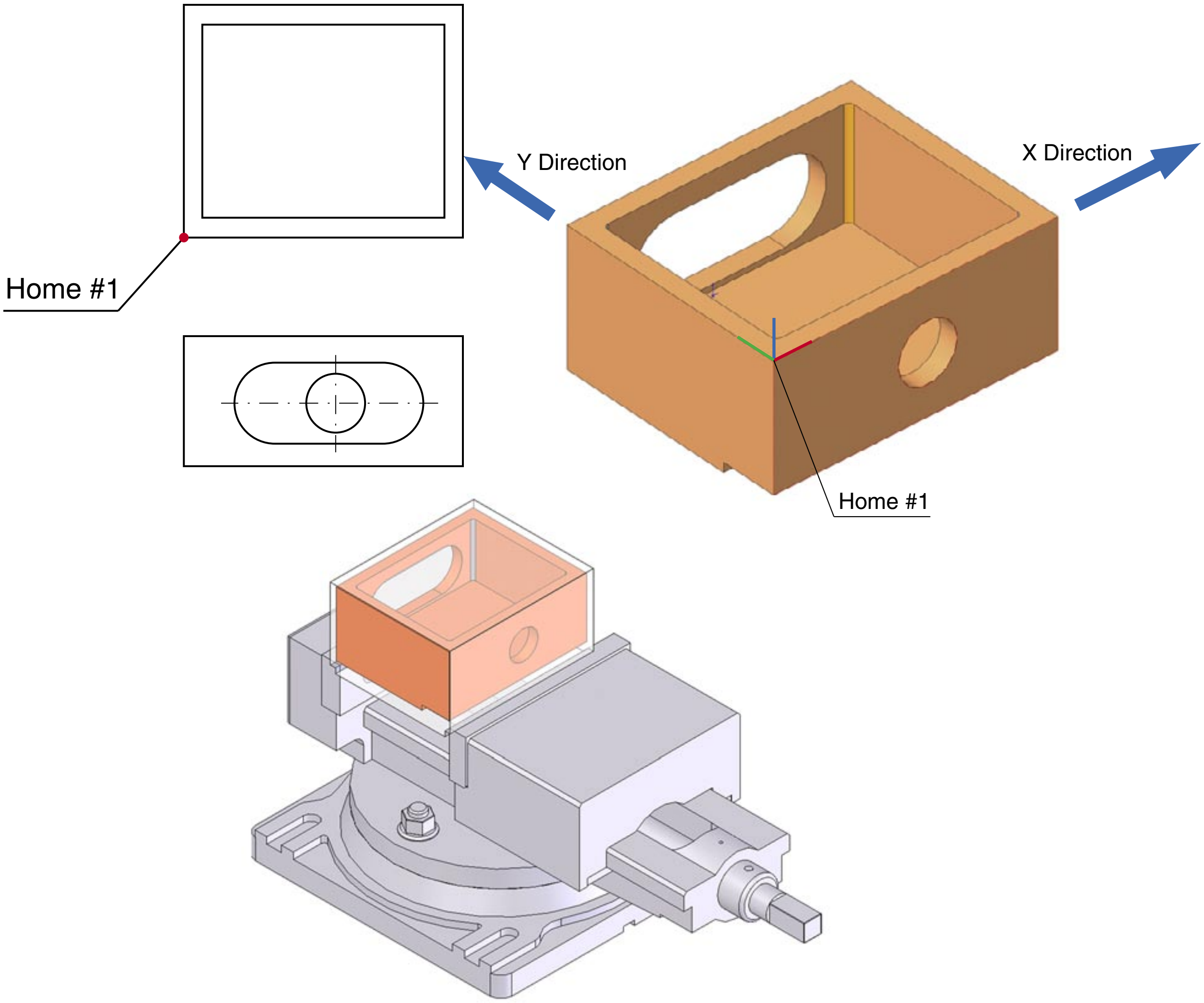


=> Pick origin point

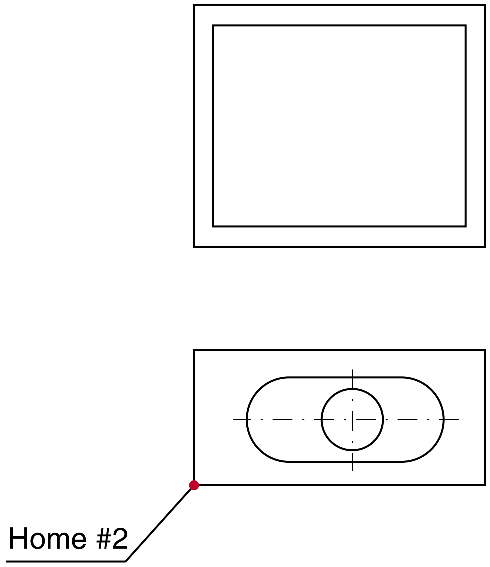
3. After you have entered the home position, you have to define the tool movement and material levels i.e. planes. The **Home data dialog** will be displayed. You can either enter the Z-level values or you can select the values from your model after clicking on the respective buttons.
4. Confirm and close the dialog by clicking the **OK** button.

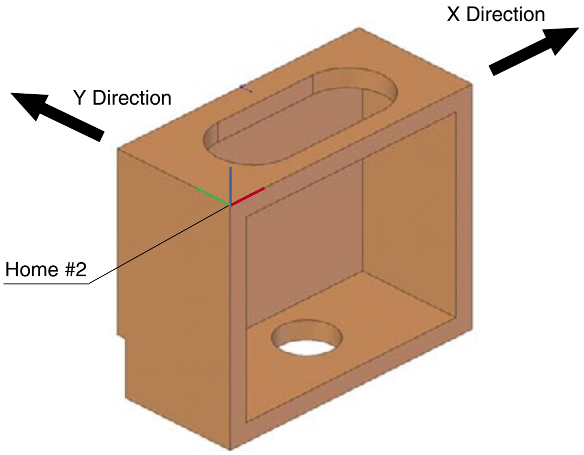
2.2.9 Defining the homes for 3-axis CNC machine within Projected mode

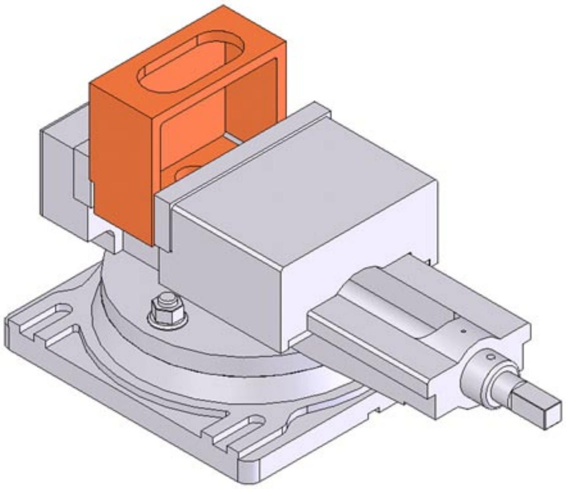
During the Home definition for 3-axis CNC machine you have to define one point for the home origin with the **Home dialog**. The point have to be defined on the top plane of the model in the orientation that will be used for machining. The X and Y axis direction will be determined automatically.



For the next home position you also have to define an origin point.



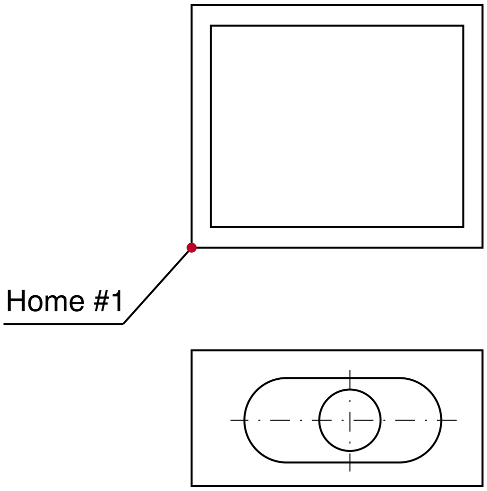

X Direction
Y Direction
Home #2

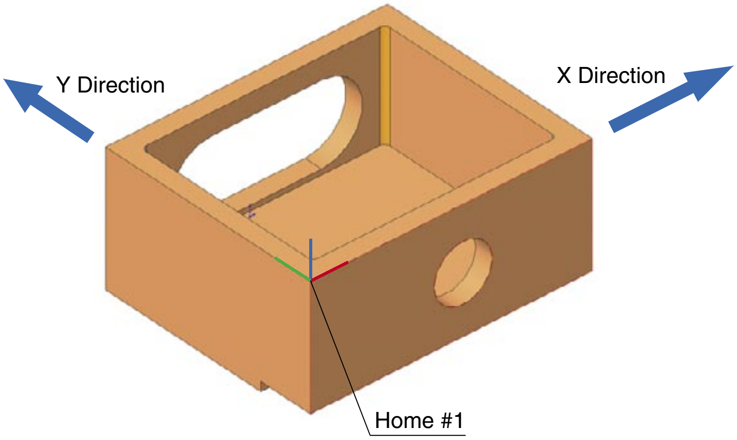


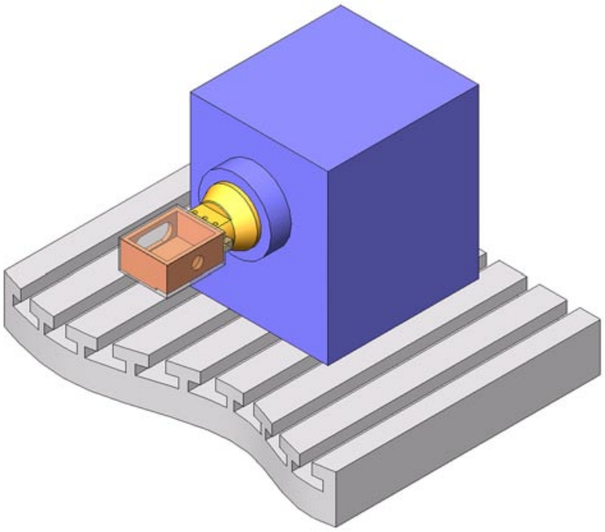
SolidCAM automatically assigns the new **Part Home number** for the each Home; the **Mac home number** can be changed by user for a new clamping (main home position).

2.2.10 Defining the homes for 4-axis CNC machine within Projected mode

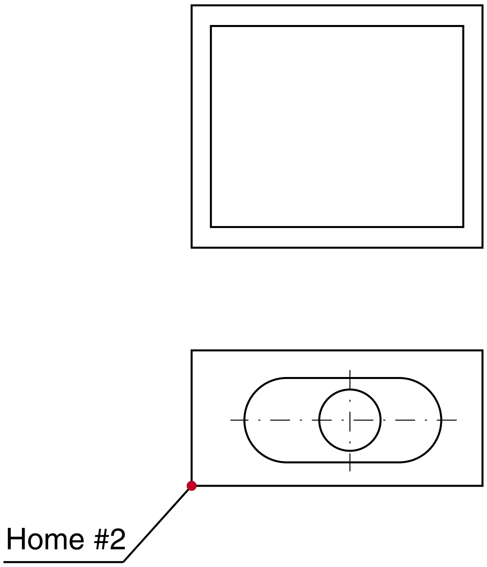
During the Home definition for 4-axis CNC machine you have to define one point for the home origin with the **Home dialog**. The point have to be defined on the top plane of the model in the orientation that will be used for machining. The X and Y axis direction will be determined automatically.

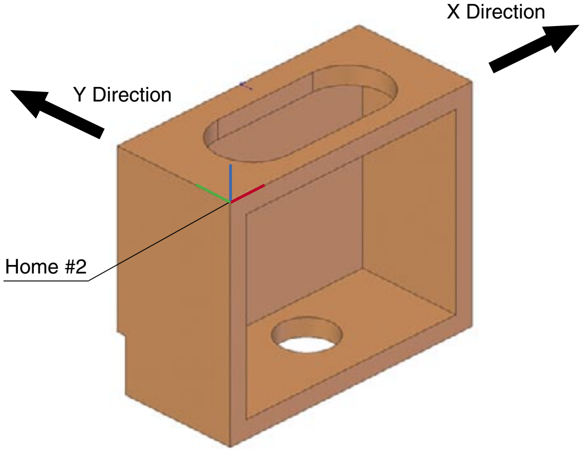


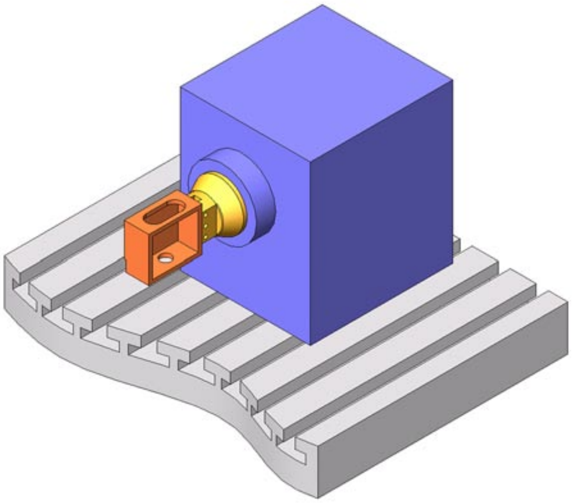




To add a new **Part Home** you have to define the point for the Home origin.





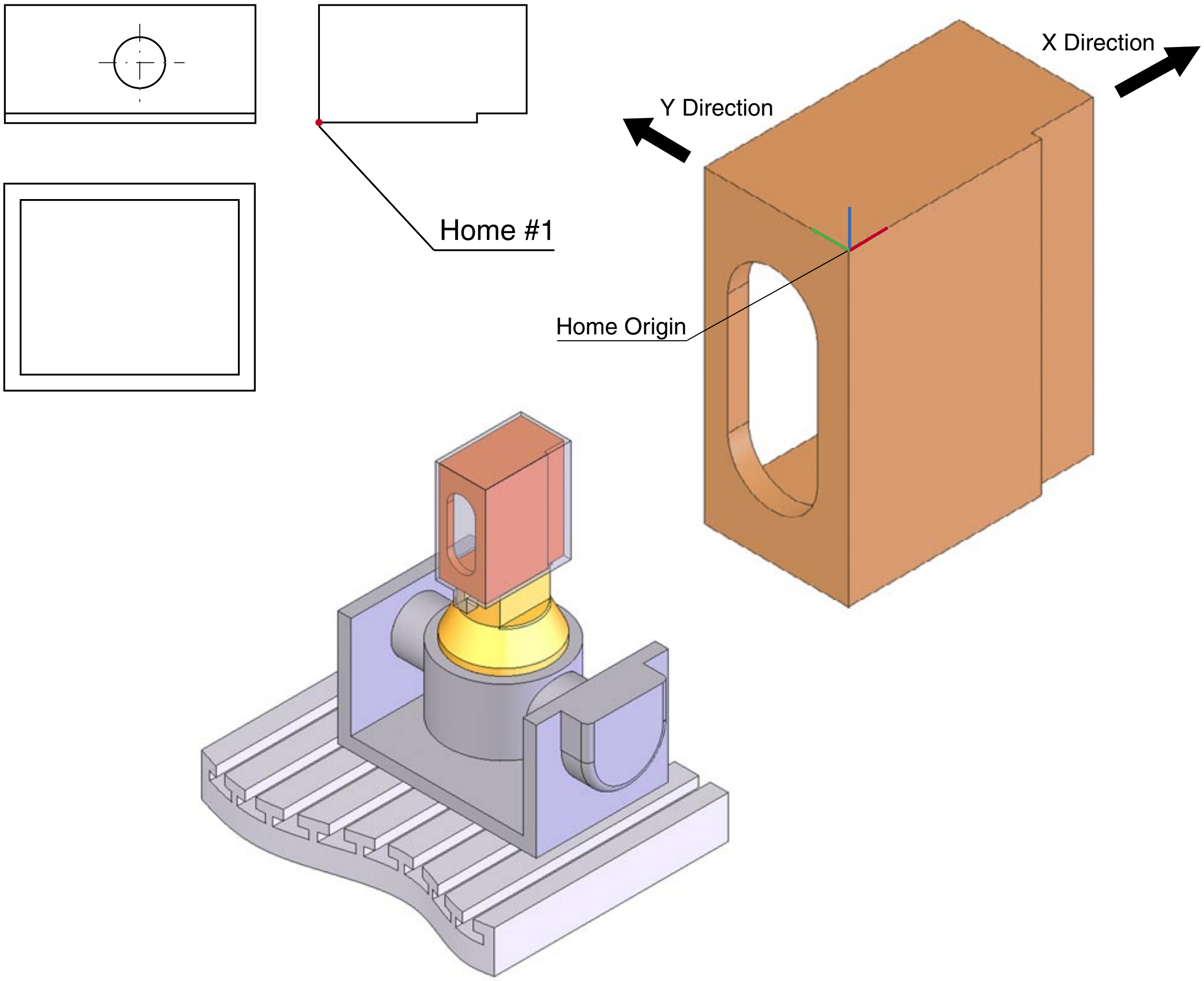


SolidCAM automatically assigns the new **Part Home number** for the each Home; the **Mac home number** can be changed by user for a new clamping (main home position).

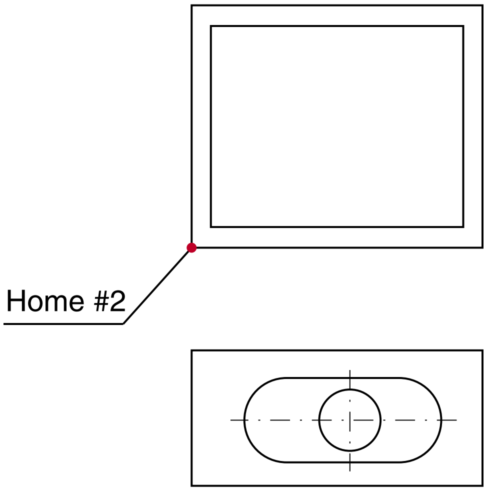
You have to define the **Shift** and **Rotation around** parameters describing position and orientation of the new Part Home according to the Mac Home.

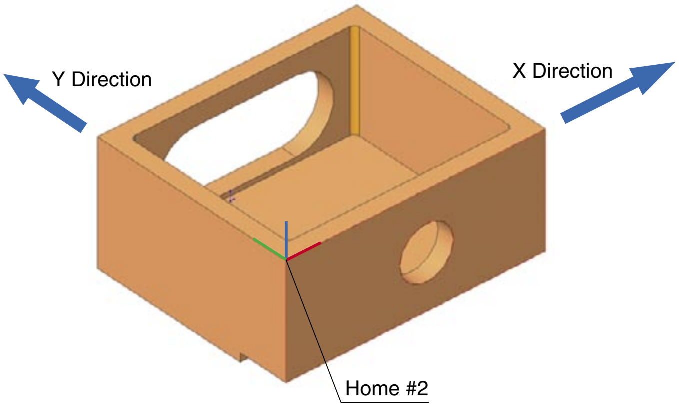
2.2.11 Defining the homes for 5-axis CNC machine within Projected mode

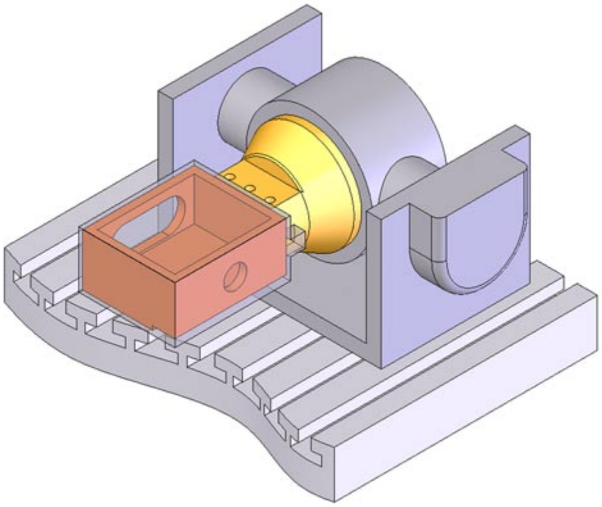
During the Home definition for 5-axis CNC machine you have to define one point for the home origin with the **Home dialog**. The point have to be defined on the top plane of the model in the orientation that will be used for machining. The X and Y axis direction will be determined automatically.



To add a new **Part Home** you have to define the point for the Home origin.





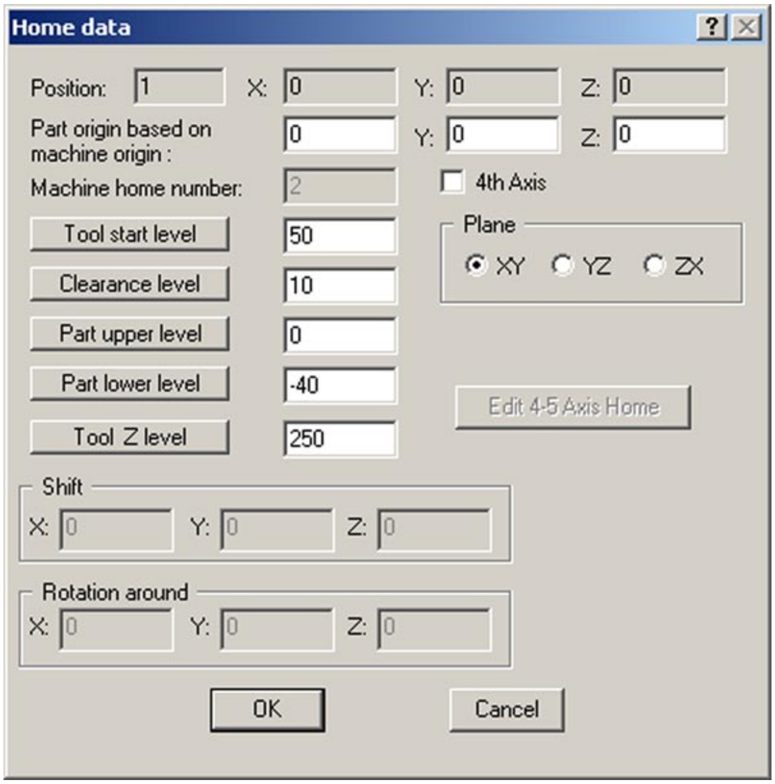


SolidCAM automatically assigns the new **Part Home number** for the each Home; the **Mac home number** can be changed by user for a new clamping (main home position).

You have to define the **Shift** and **Rotation around** parameters describing position and orientation of the new Part Home according to the Mac Home.

2.2.12 Home data dialog

Below is an explanation of the fields of the **Home data dialog**.


Home data
? ×
Tool start level
Clearance level
Part upper level
Part lower level
Tool Z level
4th Axis
Plane
XY
YZ
ZX
Edit 4-5 Axis Home
Shift
Rotation around
OK
Cancel

Position: 1 X: 0 Y: 0 Z: 0

Part origin based on machine origin : 0 Y: 0 Z: 0

Machine home number: 2

☐ 4th Axis

Tool start level: 50

Clearance level: 10

Part upper level: 0

Part lower level: -40

Tool Z level: 250

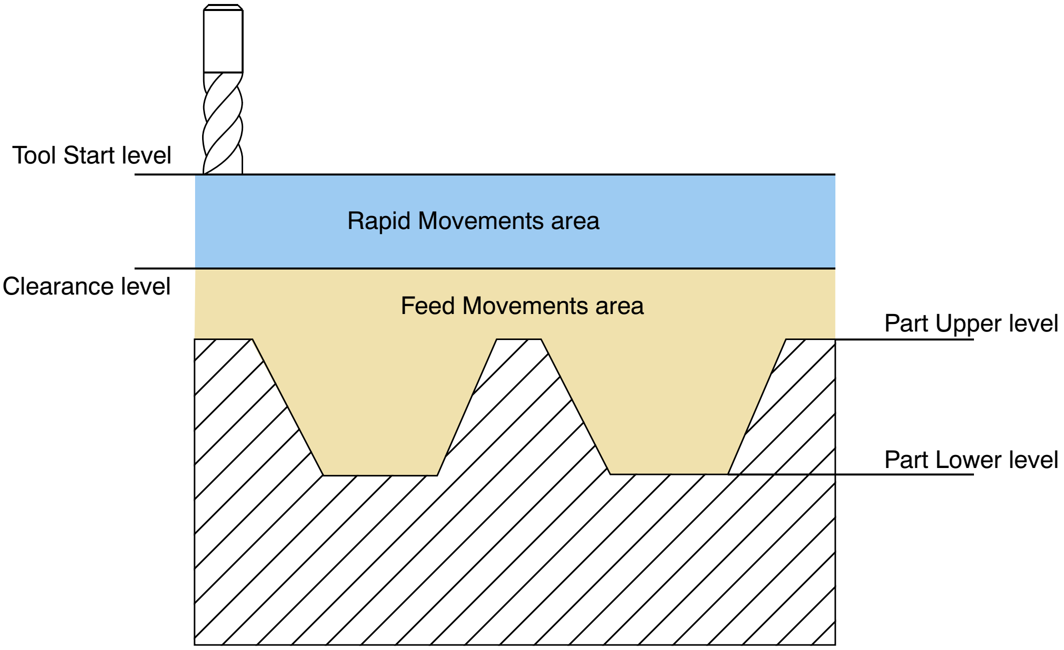
Plane: ● XY ○ YZ ○ ZX

Shift X: 0 Y: 0 Z: 0

Rotation around X: 0 Y: 0 Z: 0

- **Position** defines the sequence number of the home. For each Machine Home, several **Positions** are defined for different positionings; each such **Position** is related to the main Home.
    - **X** shows the X-value of the Home.
    - **Y** shows the Y-value of the Home.
    - Z shows the Z-value of the Home.
- **Machine home number** defines the number of the home in the CNC-Machine. The default value is 1. If you use another number, the G-Code file will contain the G-function that tells the machine to use the specified number stored in the machine controller of your machine.
- The **Tool start level** defines the Z-level at which the tool will start.
- The **Clearance level** is the Z-level to which the tool rapids when moving from one job to another (in case the tool did not change).
- **CAM-Part Upper level** defines the height of the upper surface of the part to be milled.
- **CAM-Part Lower level** defines the lower surface level of the part to be milled.

- **Tool Z Level** is the height that the tool moves to before the rotation of the 4/5 axes to avoid collision between the tool and the workpiece. This level is related to the home position and you have to check if it is not over the limit switch of the machine. It is highly recommended to send the tool to the reference point or to a point related to the reference point.



- **Shift** is the distance from the main home (Mac home) to the location of this home in the coordinate system and the orientation of the main home.

- **Rotation** is the rotation angle around the main axes X, Y and Z.

  - For the first **Part Home number** related to a new **Mac Home number**, the **Shift** and **Rotation** around parameters will be always 0.

  - **Shift** and **Rotation** for other **Part Home Numbers** related to the same **Mac Home number** will be determined automatically within **Multi-Sided mode**. Within **Projections mode** the **Shift** and **Rotation** values must be defined manually.

- The **4-th axis** field can be either **On** or **Off**. Click the mouse on this field to choose On. The 4th-Axis will then be used when changing from the previous home to the current home.

The **Machining Plane** defines the default plane of work for the jobs using this home, as it is output to the G-Code program. In the SolidCAM CAM module, you must always work on the XY-plane. Some CNC-machines, however, have different axes-definitions and require G-Code output with rotated XY-planes. Choose one of the following options:

- **XY** - The XY plane is the work plane; the Z-axis is the depth (G17).

- **YZ** - The YZ plane is the work plane; the X-axis is the depth (G18).

- **XZ** - The XZ plane is the work plane; the Y-axis is the depth (G19).

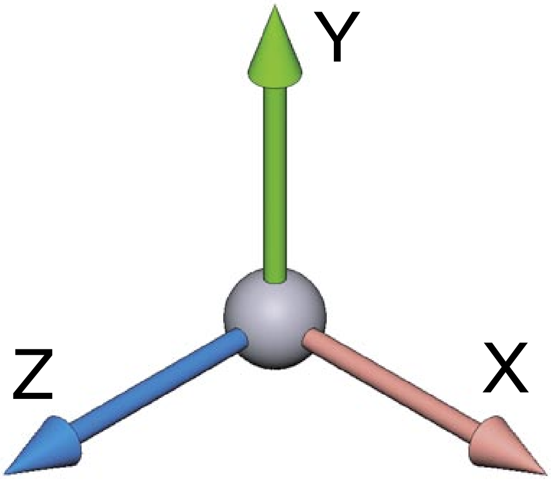
This option will only work correctly when your post-processor has been customized to support this function. Please contact **customer support** for further details.

Edit 4-5-Axis Home

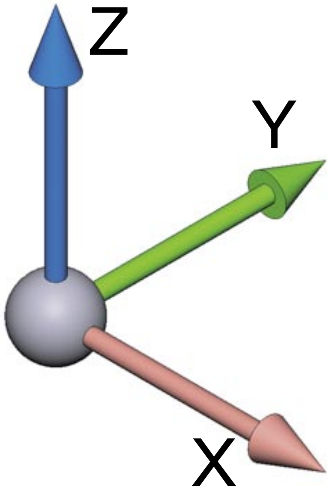
This button enables you to change the home location or orientation of axis with the **Home 4-5 Axis dialog**.

2.2.13 SolidCAM coordinate system

SolidCAM changes the model orientation by setting the model to the **Machine Isometric view**. This orientation is suitable for machining on a CNC machine (with the Z-axis pointing vertically upwards).

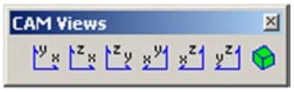


SolidWorks Coordinate system



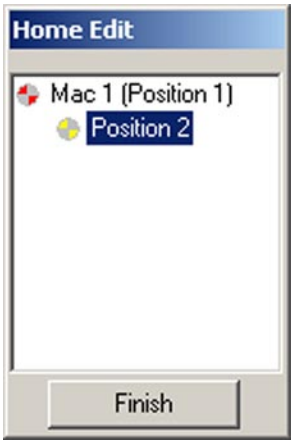
SolidCAM Coordinate system

Use the CAM Views toolbar to see the SolidCAM coordinate system. These views are related to the orientation of the selected home position.

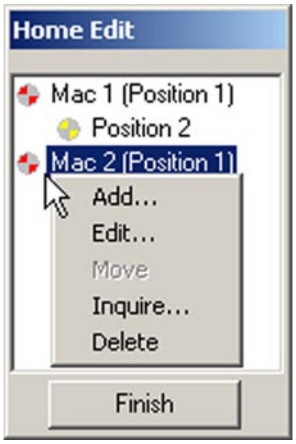


2.3 Home operations

The Home commands are available in the **CAM-Part Data dialog** after the first Home is defined. The **Home Edit dialog** will be displayed after selecting the **Define Home** button.


Home Edit
Mac 1 (Position 1)
Position 2
Finish

1. Right click on the Home name in the **Home Edit dialog**. The pulldown menu will be displayed.


Home Edit
Mac 1 (Position 1)
Position 2
Mac 2 (Position 1)
Add...
Edit...
Move
Inquire...
Delete
Finish

2. Choose the specific command from the menu.

Add Home

This commands enables you to add a new home to the current CAM-Part.

Edit Home

This command enables you to change the data of an existing home. It also enables you to change the home position.

1. **Pick home** - Choose the home you want to edit.
2. The **Home data dialog** box will be displayed showing the default home position values. The fields and values can be changed, if necessary.

If you change the home position, you must recalculate the jobs using this home position. You are also advised to review the simulation.

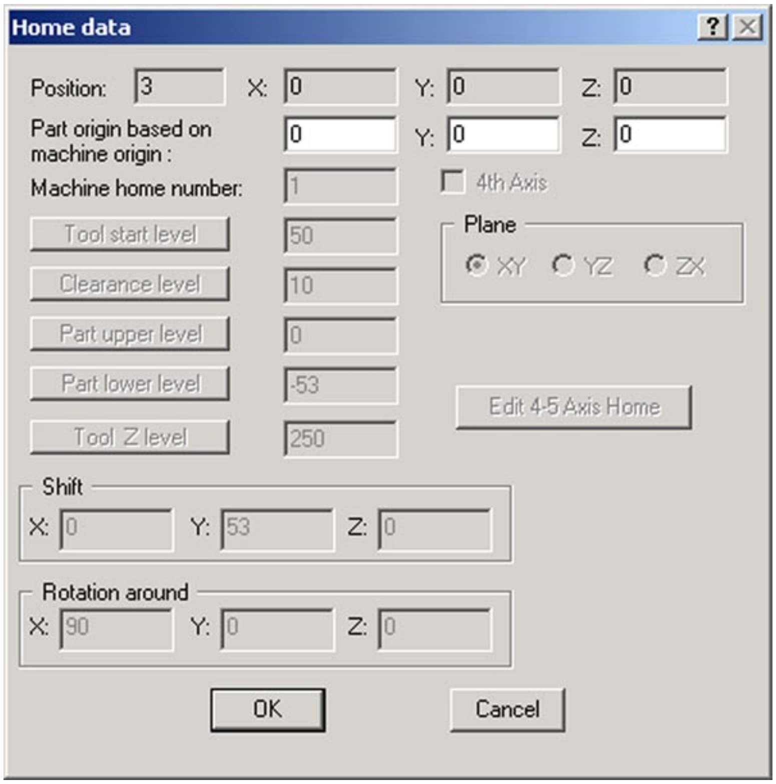
Move Home

This button enables you to move the actual Home. This option is disabled in the **Multi-sided mode**.

1. **Pick home** - Choose the home you want to move.
2. **Pick new home position** - Pick a point for the new origin.

Inquire Home

In the **Inquire Home** field you can view the relevant information about a particular home position but you cannot change anything.



Home data

Position: 3 X: 0 Y: 0 Z: 0

Part origin based on machine origin : 0 Y: 0 Z: 0

Machine home number: 1

☐ 4th Axis

Tool start level: 50

Clearance level: 10

Part upper level: 0

Part lower level: -53

Tool Z level: 250

Plane: ● XY ○ YZ ○ ZX

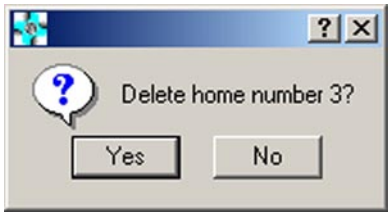
Shift: X: 0 Y: 53 Z: 0

Rotation around: X: 90 Y: 0 Z: 0

Delete Home

Deletes an existing home. You can only delete homes if more than one home has been defined for the current CAM-Part. It is not possible to delete the first home and a home that is used for the milling operation. (The **Delete** item in the menu will be вdisabled in these cases).

1. **Pick home** - Choose the home you want to delete. A confirmation message will be displayed.

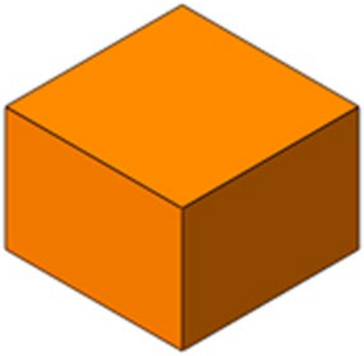


Delete home number 3?

2. Choose **Yes** to confirm.

2.4 Stock and Target Model

Stock model - the material that is placed on the machine before you start machining the CAM-Part.

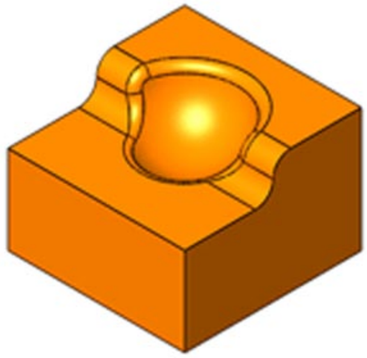


The **Stock model** is used for:

- Solid Verify and VerifyPlus simulation.
- For rest material calculation in 3D jobs, in case the field Update Stock model is checked.

The stock model is not automatically associated with the solid model. If the model is changed and the stock is not correct anymore you will see it in the simulation; you can then replace the defined stock with the updated one.

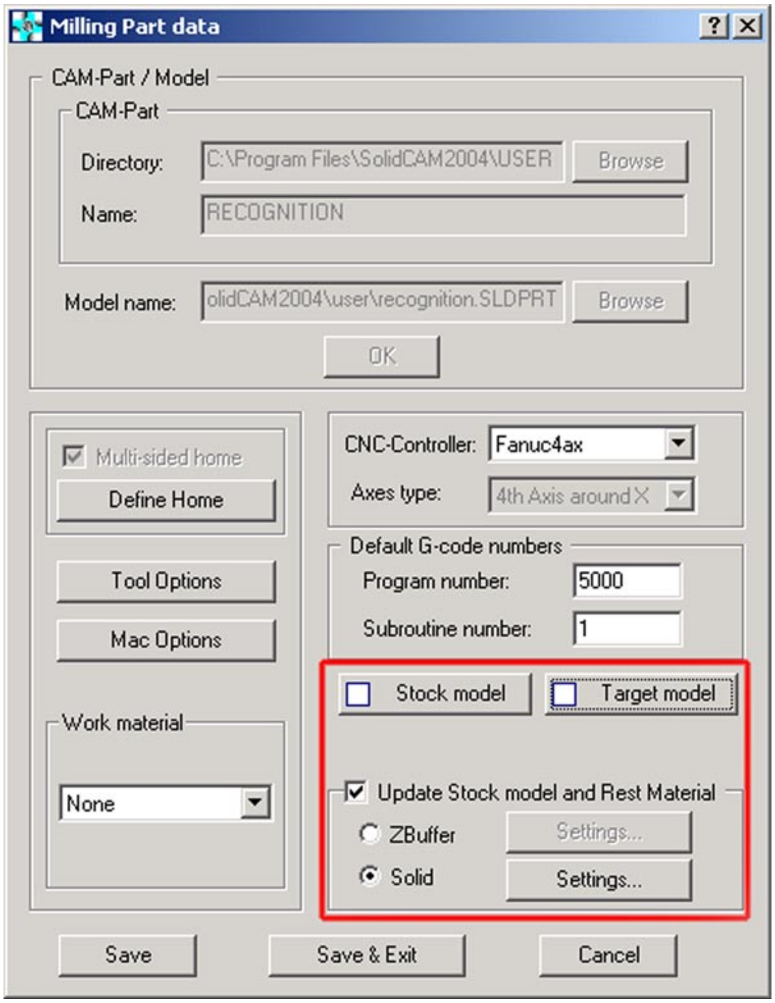
Target model - the final shape of the CAM-Part after the machining.



SolidCAM will use the **Target model** for:

- 3D simulation
- Rest material calculation
- Gouge checking in SolidVerify simulation.

During all machining operations, the **rest material** that is left is removed in order to reach the Target model.



Milling Part data

CAM-Part / Model

CAM-Part

Directory: C:\Program Files\SolidCAM2004\USER

Name: RECOGNITION

Model name: olidCAM2004\user\recognition.SLDPRT

☑ Multi-sided home

CNC-Controller: Fanuc4ax

Axes type: 4th Axis around X

Default G-code numbers

Program number: 5000

Subroutine number: 1

☐ Stock model ☐ Target model

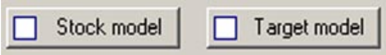
☑ Update Stock model and Rest Material

○ ZBuffer

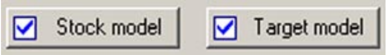
● Solid

Work material: None

Browse
Browse
OK
Define Home
Tool Options
Mac Options
Settings...
Settings...
Save
Save & Exit
Cancel


☐ Stock model
☐ Target model

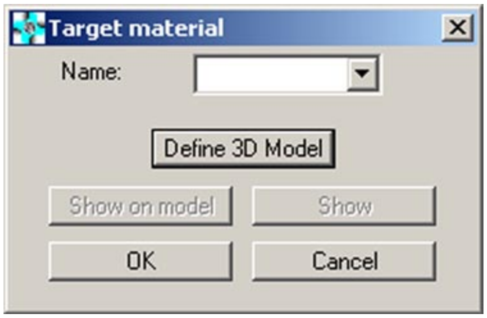
If the small square is unchecked, no stock & target models have been defined.


☑ Stock model
☑ Target model

If the small square is checked, the stock & target models have been defined.

2.4.1 Target model definition

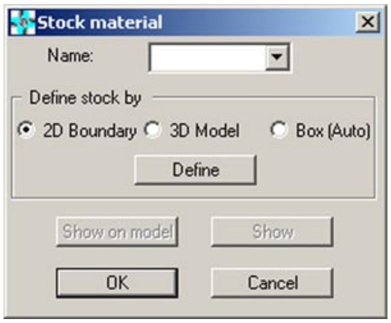
The **Target model** option enables you to define the final part after machining.


Target material
Name:
Define 3D Model
Show on model
Show
OK
Cancel

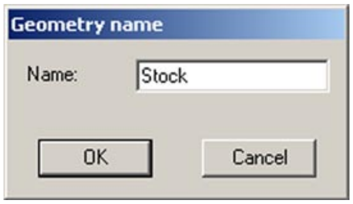
1. Click on the **Target model** button in the **CAM-Part data dialog**
2. Click on the **Define 3D model** button to start the geometry selection.
3. Select the geometry with the **3D geometry dialog** (see the topic **4.4.1**).
4. After the geometry selection you will be prompted to enter the target model name. Enter a name and click on the **OK** button. In future 3D operations you can use the Target material as the defined geometry.

2.4.2 Stock Model

The **Stock model** option enables you to see the actual material that has to be removed.


Stock material
Name:
Define stock by
2D Boundary
3D Model
Box (Auto)
Define
Show on model
Show
OK
Cancel

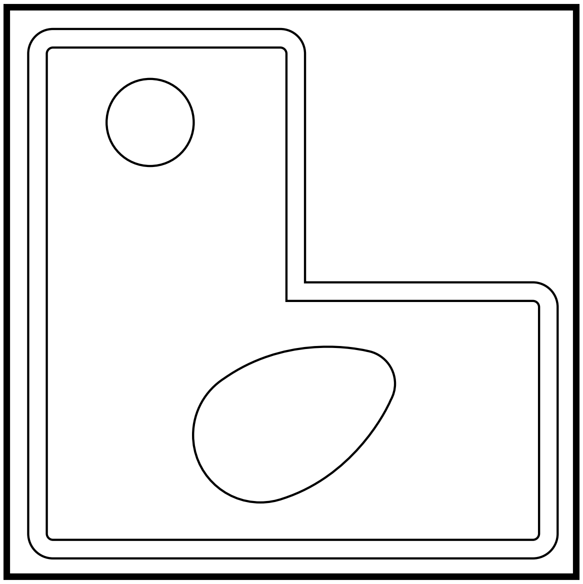
1. Click on the **Stock model** button in the **CAM-Part Data dialog**.
2. Choose the stock model definition **mode** and click on the **Define** button.
3. Select the geometry.
4. After geometry selection you will be prompted to enter the stock model name. Enter a name and click on the **OK** button.


Geometry name
OK
Cancel

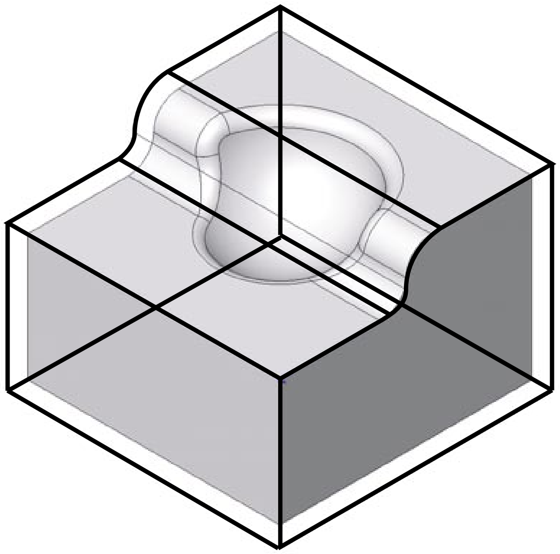
Name: Stock

Modes

- **2D Boundary** - this mode enables you to define the 2D stock geometry with the chain of the geometry elements (lines, arcs, splines, edges... etc). Before choosing the stock material, you first have to draw the boundaries of the stock material around the CAM-Part. After choosing this mode and pressing the define button, the **Geometry Edit dialog** will be displayed.

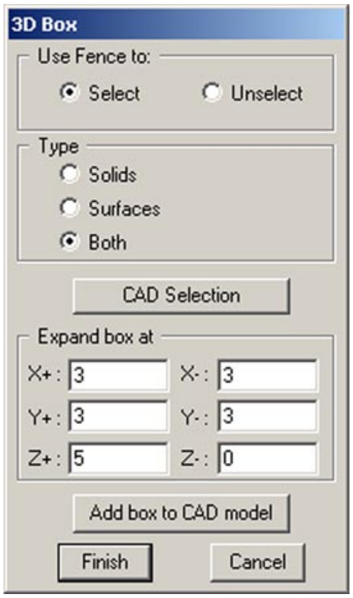


- **3D Model** - this option enables you to define the stock model via 3D model selection. This option is useful for casting machining. After choosing this mode and pressing the define button, the **3D geometry dialog** will be displayed.



- **Box (Auto)** - In this case SolidCAM automatically finds the box surrounding the model. After pressing the **Define** button, the **3D Box dialog** will be displayed.

The **3D box dialog** is used to select a 3D model to create a 3D box around the model.



3D Box

Use Fence to:
- ● Select
- ○ Unselect

Type
- ○ Solids
- ○ Surfaces
- ● Both

Expand box at
X+ : 3
X- : 3
Y+ : 3
Y- : 3
Z+ : 5
Z- : 0

Use Fence to:

This option enables you to choose what to do with the fence that you dragged over the part.

- **Select** - Select is used to choose the model with the fence.
- **Unselect** - Unselect is used to remove a model with the fence that was already chosen.

Type

This option enables you to choose the type of elements from the model to use.

- **Solids** - Allows solid models to be included in the fence.
- **Surfaces** - Allows surfaces to be included in the fence.
- **Both** - Allows both solid models and surfaces to be included in the fence.

CAD Selection

Enables you to pick the model with the Host CAD tools.

Expand box at

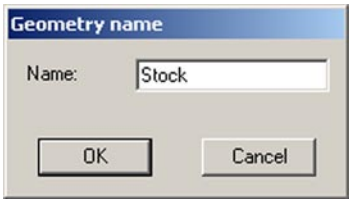
This field gives you complete control over how much access stock material you have in each direction of the stock material.

Add box to CAD model

This option enables you to add the stock material to the CAD model.

Finish

Pressing **Finish** will close this field. The **Geometry Name** dialog will be displayed.



Name: Stock

Enter a name and press **OK**.

Pressing **OK** will close this field and the **CAM-Part Data** field will be displayed.

Common controls for Target Model and Stock model dialogs

Name

In this field you can see the target or stock model name and choose the model from the previously defined 3D geometries.

Show

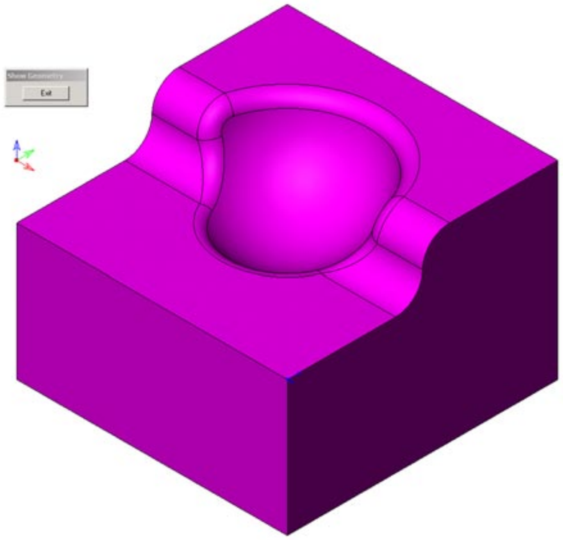
With this button the Stock or Target Model will be shown in the **Rest Material** window of SolidCAM.

If you want to control the quality of the stock & target models visualization, you can change the target tolerance in the **SolidCAM Settings (**see the topic **10.3.4)**.

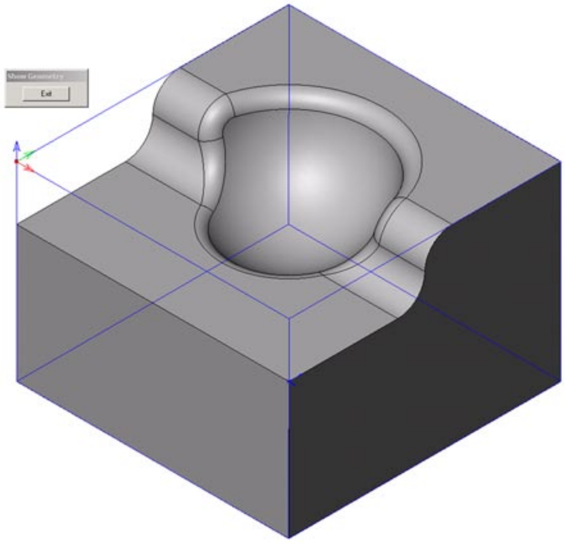
- In the SolidCAM main menu press the **CAM settings…**
- In the **Units** tab you can change the **Target tolerance**. The recommended tolerance is **0.1mm**. A smaller tolerance will improve the quality and reduce the visualization speed.

Show on model

With this button you can see the selected Target model in the **Host CAD** window.



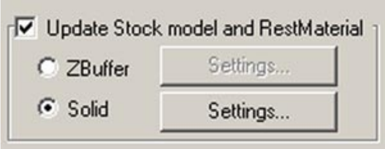
Target Model



Stock model

2.4.3 Updating Stock Model

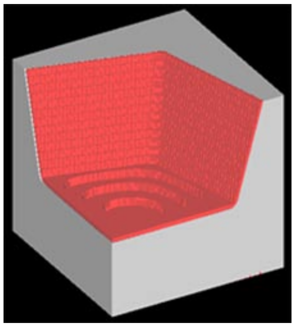
Update Stock model - After every Job, SolidCAM calculates how much material was actually removed from the CAM-Part and how much material is left not machined.



☑ Update Stock model and RestMaterial
○ ZBuffer
● Solid

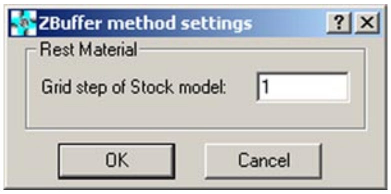
Z Buffer

With this mode, the rest material is calculated only for one machining direction.



The red area is the rest material left unmachined after the roughing.

Settings - Pressing the **Settings** button will open a field that enables you to control the grid step for the stock model triangulation.



ZBuffer method settings

Rest Material

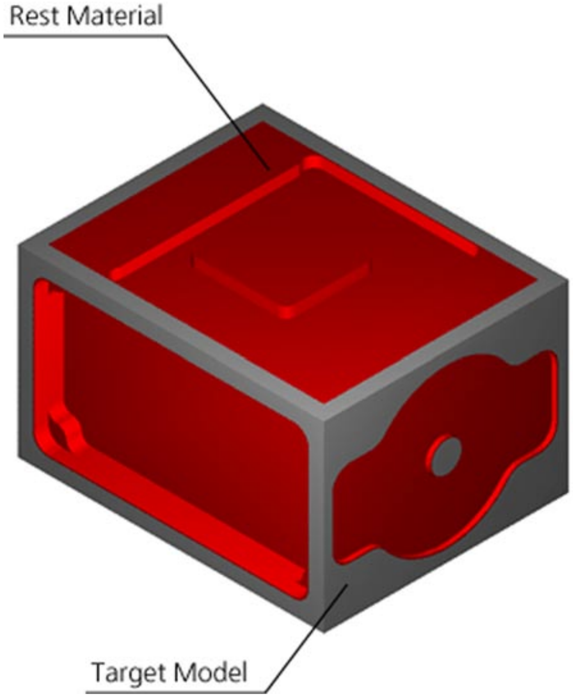
Grid step of Stock model: 1

The smaller the grid step, the more precise is the stock model update after each job. However the smaller the grid step, the greater is the calculation time for updating the stock model. It is recommended not to change the default value calculated by SolidCAM, relative to the stock model size.

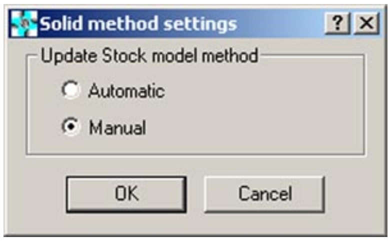
After every job calculation, SolidCAM will update the stock model and save the volume removed by the job tool path.

Solid

This mode enables you to calculate the Rest material from all sides.



Settings - Pressing the **Settings** button will open the Solid method settings dialog. This dialog enables you to choose either **Automatic** or **Manual** method of the Rest material calculation.



○ Automatic
● Manual

- **Automatic method**

  This option causes SolidCAM to automatically calculate and save the rest material after each job. During the **SolidVerify simulation** (see the topic **8.6**) of the separate jobs, SolidCAM updates the stock with the previously generated and saved rest material and performs the simulation on the updated model. Since the rest material models are saved after each job calculation, SolidCAM enables you to simulate any job for multi-sided CAM-Parts.

- **Manual method**

  SolidCAM enables you to manually save the Rest Material for a specific job during the **SolidVerify simulation** (see the topic **8.6**). The saved model can be used to update the stock. The simulation of the next jobs can be performed on the updated model.

2.5 Tool options

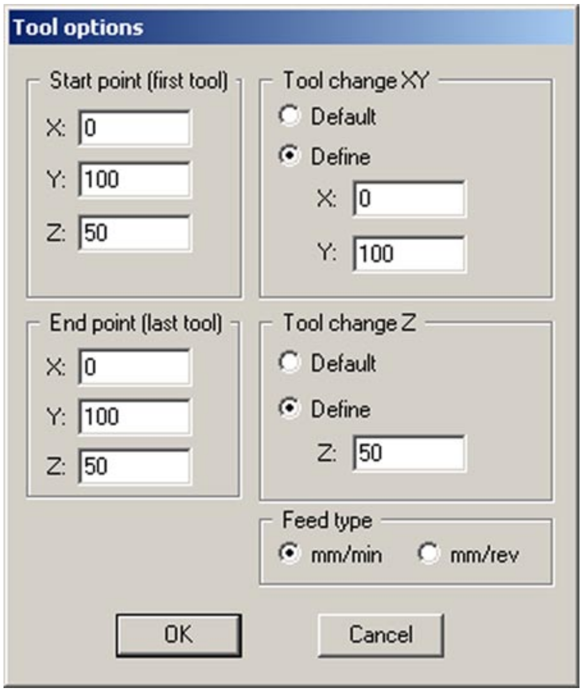
The Tool options enable you to define additional data about the tools for your CAM-Part. You can enter the start and end position of your tool as well as the coordinates for tool changes on your CNC-machine. After you click on the button, you can change the parameters in the **Tool option dialog** box.

The default values you will receive in this dialog are input from the *.mac file of the selected post-processor.

A sample from the *.mac file is shown below, where the first three values of each line are metric coordinates and the last three values are in inches.

```
;Positioning

dflt_start = 0.0000 200.0000 100.0000, 0.0000 7.8740 3.9370

dflt_end = 0.0000 200.0000 0.0000, 0.0000 7.8740 0.0000

set_z_chng = Y

dflt_tool_chng = 0.0000 0.0000 0.0000, 0.0000 0.0000 0.0000
```

In the **Tool options dialog** you can edit the values for the current CAM-Part.



Start point (first tool): X: 0, Y: 100, Z: 50

End point (last tool): X: 0, Y: 100, Z: 50

Tool change XY: ○ Default ● Define X: 0 Y: 100

Tool change Z: ○ Default ● Define Z: 50

Feed type: ● mm/min ○ mm/rev

Start Point

Determines the position of the first tool at the start of the milling.

End Point

Determines the position of the last tool at the end of the milling.

- In most of the cases the value in this dialog will only affect the simulation. In order for this to affect the generated NC program, the postprocessor must use these values.
- Normally the new CNC controllers do not need the start and end position. In the postprocessor we use the machine command to move to the machine reference point.

Tool Change XY

Defines the XY coordinates of the tool change position. It has 2 possibilities:

- **Default as in *.mac file**

  The XY coordinates of the tool change position are defined by the end point of the job that was executed before the tool change.

- **Define**

  The XY coordinates of the tool-change position are defined by the user.

Tool Change Z

Defines the Z coordinate of the tool-change position. It has two possibilities:

- **Default as in *.mac file**

  The Z-level where tool changes are performed, is automatically read from the Tool start plane specified in the Home data dialog (default input from *.mac file).

- **Define**

  The Z-coordinate of the tool change position is defined by the user.

Feed Type

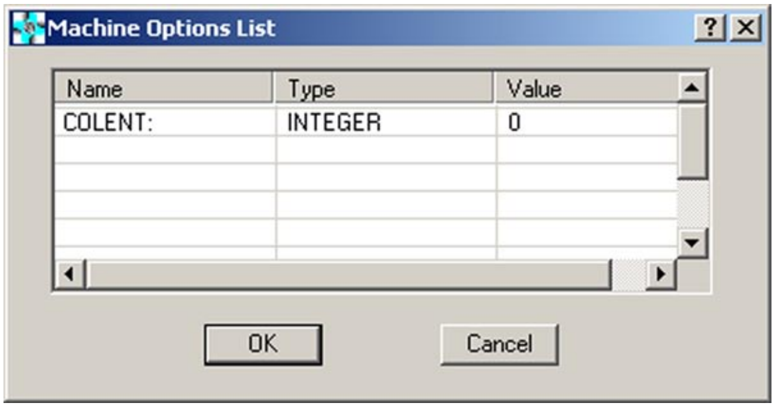
Defines the default type of feed used for the tool of this CAM-Part. It has two options:

- **Mm/Min** - Millimeters per Minute
- **Mm/Rev** - Millimeters per Revolution

2.6 Mac Options

If the ***.mac** file of your post-processor has extra fields (parameters) defined, they can be accessed through this field. In the **Mac Options dialog** you can insert unique information related to the machine controller you have selected.

- In case of **Numeric** or **Integer** field type, the **Numeric** / **Integer Value dialog** box is displayed and you are prompted to enter a value.
- In case of **Logical** field type, the **Logical Value dialog** box is displayed and you are prompted to choose between **Yes** and **No**.

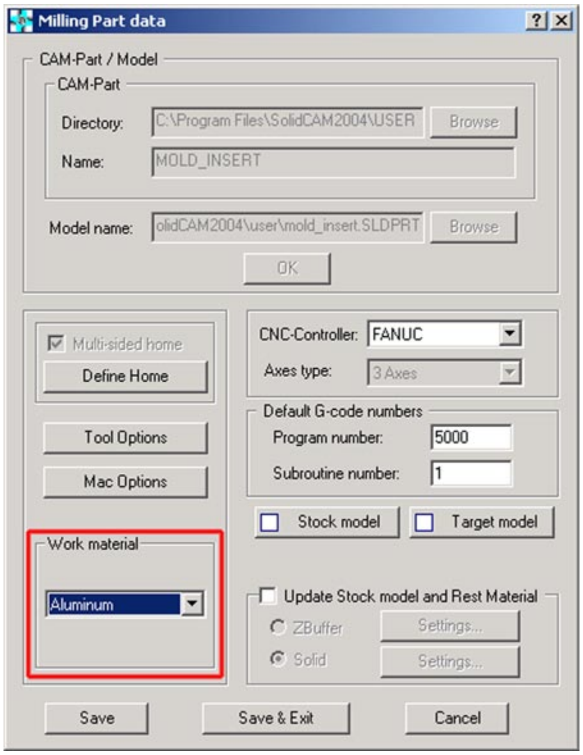


Name	Type	Value
COLENT:	INTEGER	0

Refer to the **GPPTOOL manual** for additional information.

2.7 Work Material

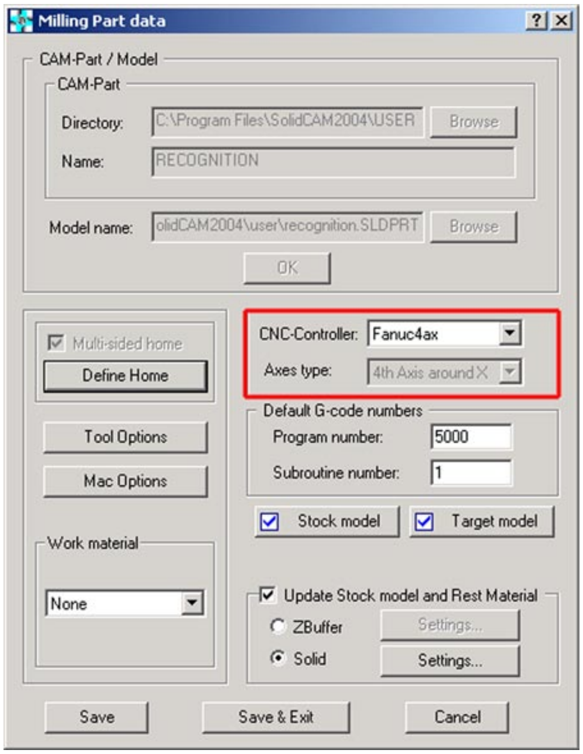
This option enables you to define the work material. Choose the **Work Material** from the list in the list.


Milling Part data
CAM-Part / Model
CAM-Part
Browse
Browse
OK
Define Home
Tool Options
Mac Options
Default G-code numbers
Stock model
Target model
Settings...
Settings...
Save
Save & Exit
Cancel

Directory: C:\Program Files\SolidCAM2004\USER
Name: MOLD_INSERT
Model name: olidCAM2004\user\mold_insert.SLDPRT
☑ Multi-sided home
CNC-Controller: FANUC
Axes type: 3 Axes
Program number: 5000
Subroutine number: 1
☐ Stock model
☐ Target model
Work material: Aluminum
☐ Update Stock model and Rest Material
○ ZBuffer
● Solid

The **Work material** enables you to get **Speed/Feed Defaults** (see the topic **3.11**) based on the work and tool materials that you are using. If you choose the **None** option the system will offer you an internal standard value spin of 1000 rpm and a default feed for X, Y movement of 100 mm/min.

2.8 CNC-Controller and Axis type



Milling Part data

CAM-Part / Model

CAM-Part

Directory: C:\Program Files\SolidCAM2004\USER

Name: RECOGNITION

Model name: olidCAM2004\user\recognition.SLDPRT

☑ Multi-sided home

CNC-Controller: Fanuc4ax

Axes type: 4th Axis around X

Default G-code numbers

Program number: 5000

Subroutine number: 1

☑ Stock model

☑ Target model

Work material: None

☑ Update Stock model and Rest Material

○ ZBuffer

● Solid

Select the **CNC Machine Controller**. Click the mouse on the arrow next to the controller field to display the list of post-processors installed on your system.

2.8.1 Axis type

This field displays the CNC-controller type. This field cannot be changed; it is defined in the postprocessor file.

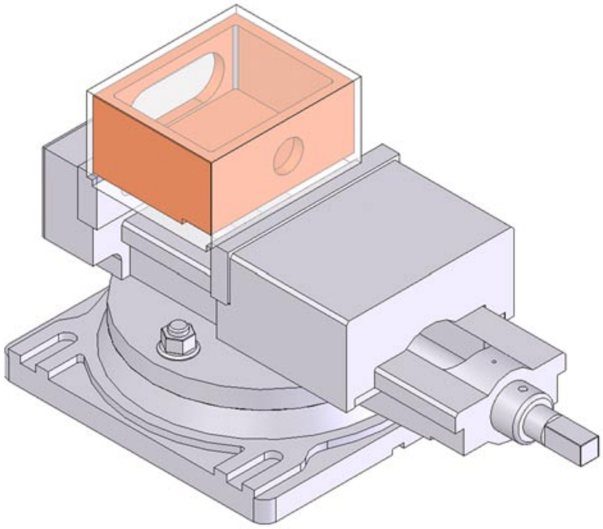
The **Axis type** can be one of the 3 following types:

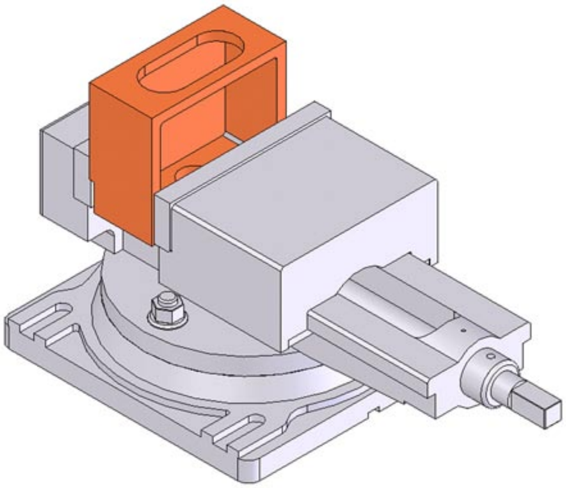
- **3 axis**
- **4_axis_around_X or 4_axis_around_Y**
- **5 axis**

The axis type will decide the home definition method:

3 axis

In this type of machine, every side requires a new clamping (new main home). For each main home you will have to define the home-origin position, X-axis direction and the Y-axis direction.



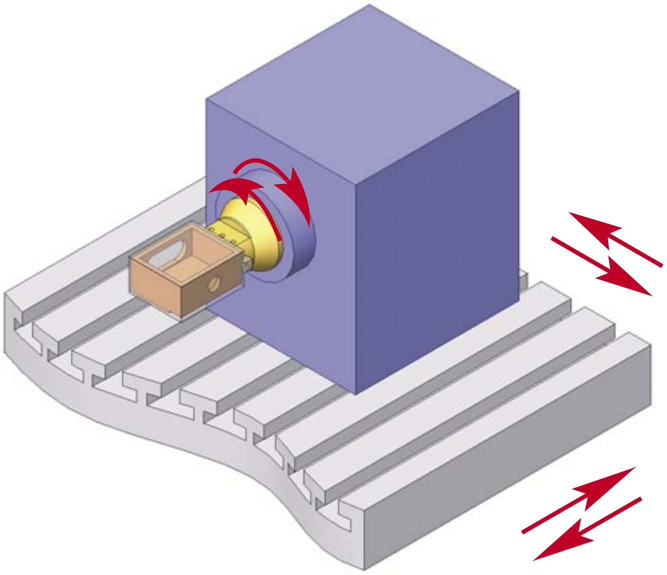


4 axis

- **4_axis_around_X**

  This type is used for vertical machines with the 4th axis on the table along the X-axis.

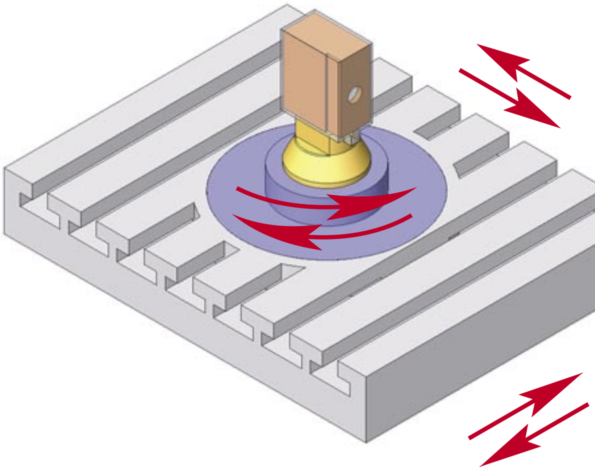
  When defining the main home you have to define the home-origin position, the X-axis direction and the Y-axis direction. For all additional positioning, based on this clamping, you have to define the home-origin position and the Y-axis direction only (the angle); the X-axis direction will be determined automatically by the X-axis direction of the main home.



- **4_axis_around_Y**

This type is used for horizontal machines with the 4th axis along the Y-axis.

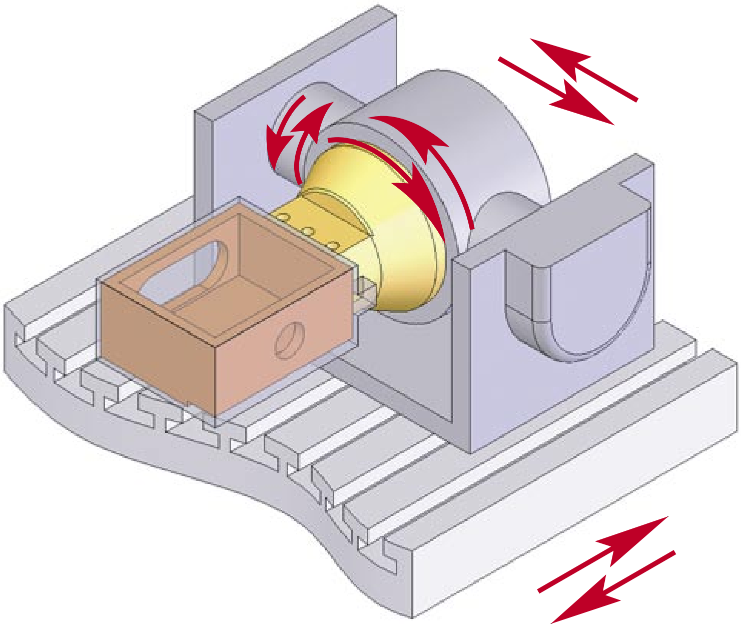
When defining the main home you will have to define the home-origin position, the X-axis direction and the Y-axis direction. For all additional positioning, based on this clamping, you have to define the home-origin position and the X-axis direction only (the angle); the Y-axis direction will by determined automatically by the Y-axis direction of the main home.



5 axis

This type is used for 5-axis CNC machines. For the main home you have to define the home-origin position, the X-axis direction and the Y-axis direction. All additional positioning is related to this home.

The user has to take care of the machine limitations (angles or undercuts). A message on the limit can be generated only during G-Code generation.



In the definition process, it is not necessary to consider the kinematics of the CNC machine. This is done in the last step when you generate the NC-program.

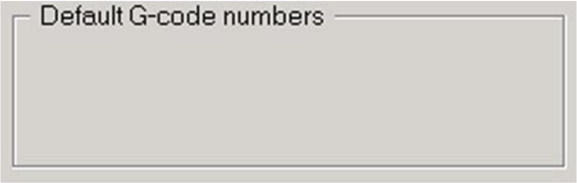
- A CAM-Part created for a CNC-controller of 3-axis type cannot be converted to 4- or 5- axis.
- A CAM-Part created for the CNC-controller of 4-axis type can be converted to 3-axis type.
- A CAM-Part created for the CNC-controller of 5-axis type can be converted to 3-axis type.

2.9 Default G-Code numbers

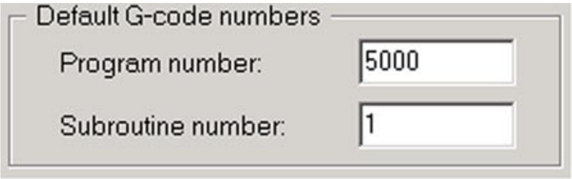
The fields in this area will be displayed according to a parameter defined in the .mac file.

```
; Program numbers
get_prog_num        = N    or    Y
get_proc_num        = N    or    Y
```

If **N** - the field will not be displayed.



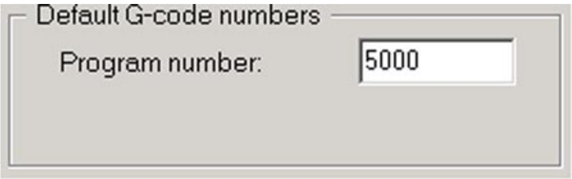
If **Y** - the field will be displayed.



Program number: 5000

Subroutine number: 1

```
get_prog_num        = Y
get_proc_num        = N
```



Program number: 5000

Program Number

This field enables you to define the number of the main program that will be generated in the output NC program.

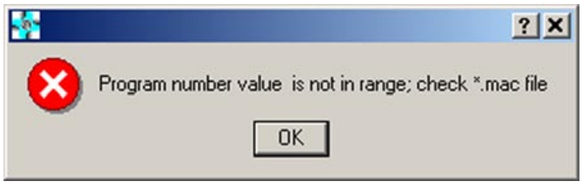
This field has ranges that are controlled from the mac file parameters.

prog_num_min = 1

prog_num_max = 9999

prog_num_dflt = 5000

In this case the default number will be 5000. You can change this value and type a number between 1 and 9999. Typing any number beyond this range will be followed with a message:



Program number value is not in range; check *.mac file

Subroutine Number

In this field you can define the number of the first subroutine in the generated output NC program.

This field has ranges that are controlled from the mac file parameters.

proc_num_min = 1000

proc_num_max = 5000

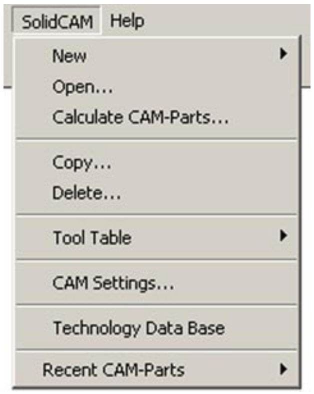
proc_num_dflt = 1000

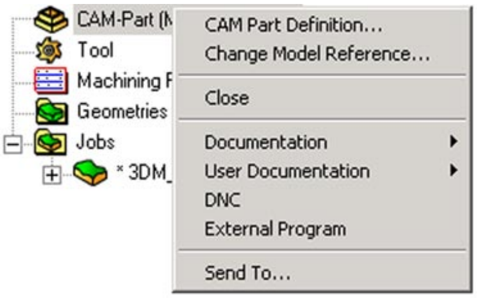
In this case the default number will be 1000. You can change this value and type a number between 1000 and 5000. Typing any number beyond this range will be followed with the same message as above.

In some machines (example: Fanuc) the main program number and subroutine number cannot be the same. SolidCAM will not warn you about this problem. In order to prevent this situation, we can set the main program range to 1 - 1000 and the subroutines range to 1001 to 9999 in the Mac file.

2.10 Managing CAM-Parts

You can use the commands of the SolidCAM and CAM-Part menus to edit and manage existing CAM-Parts.

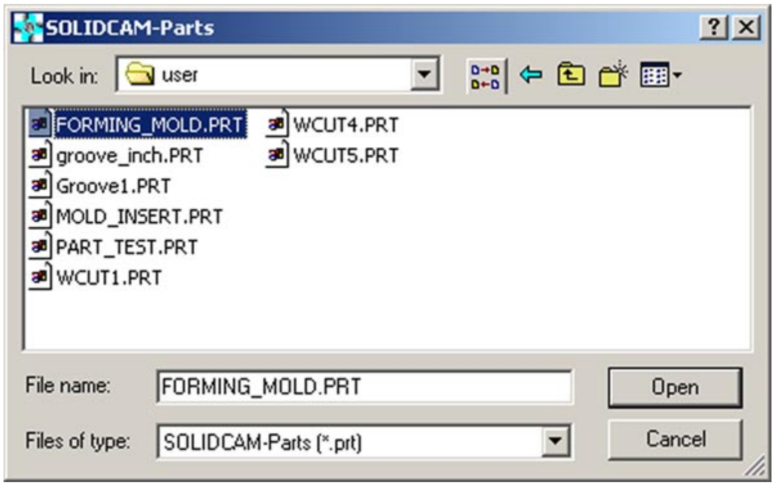

SolidCAM
Help
New
Open...
Calculate CAM-Parts...
Copy...
Delete...
Tool Table
CAM Settings...
Technology Data Base
Recent CAM-Parts


CAM-Part (N
Tool
Machining F
Geometries
Jobs
* 3DM_
CAM Part Definition...
Change Model Reference...
Close
Documentation
User Documentation
DNC
External Program
Send To...

Open

This command loads a CAM-Part that has been previously saved. When you choose this option, the **CAM-Parts browser** will be displayed.

1. Select the CAM-Part you want to load.


SOLIDCAM-Parts
Look in:
user
FORMING_MOLD.PRT
groove_inch.PRT
Groove1.PRT
MOLD_INSERT.PRT
PART_TEST.PRT
WCUT1.PRT
WCUT4.PRT
WCUT5.PRT
Open
Cancel

File name: FORMING_MOLD.PRT
Files of type: SOLIDCAM-Parts (*.prt)

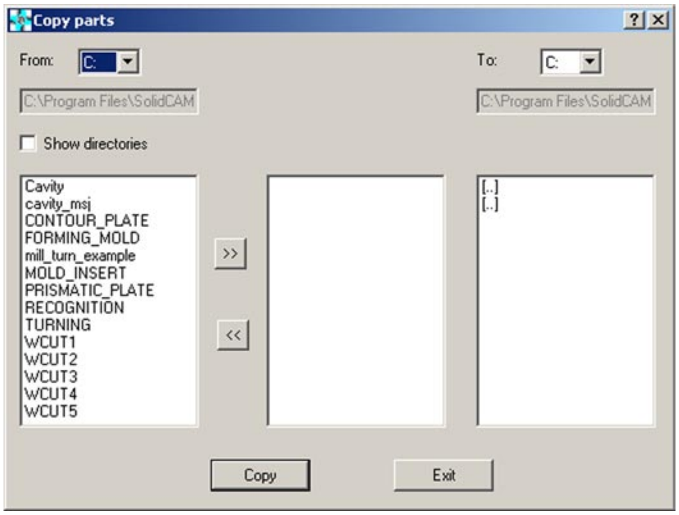
2. Double click on the CAM-Part or select it and click on the **Open** button.

Close

Closes the current CAM-Part.

Copy

The **Copy** command creates an identical copy of a CAM-Part in another folder or drive for backup purposes or for making modifications without changing the original. The **Copy CAM-Parts dialog** will be displayed.


Copy parts
Show directories
Cavity
cavity_msj
CONTOUR_PLATE
FORMING_MOLD
mill_turn_example
MOLD_INSERT
PRISMATIC_PLATE
RECOGNITION
TURNING
WCUT1
WCUT2
WCUT3
WCUT4
WCUT5
>>
<<
[..]
[..]
Copy
Exit

From: C:

C:\Program Files\SolidCAM

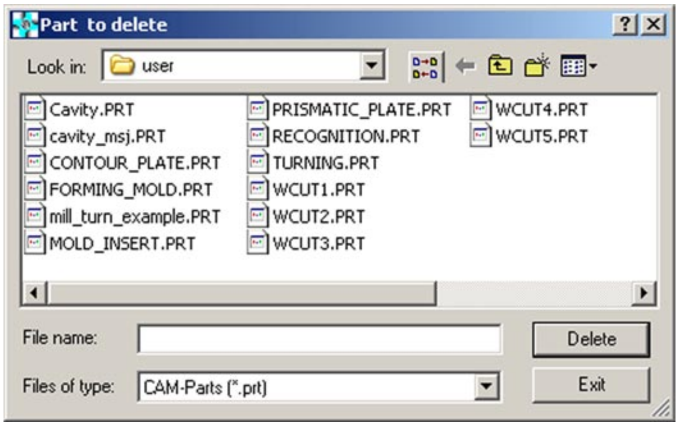
To: C:

C:\Program Files\SolidCAM

1. Choose the drive from which you want to copy the original CAM-Part.
2. Choose the CAM-Parts you want to copy from the CAM-Parts list.
3. Transfer the list of the CAM-Parts you want to copy to the middle field by pressing the Left to Right arrow.
4. Choose the drive and the directory you want to copy to.
5. Press the **Copy** button in order to execute the **Copy** command.

Delete

This field enables you to delete a CAM-Part. The **Delete CAM-Parts dialog** box will be displayed.

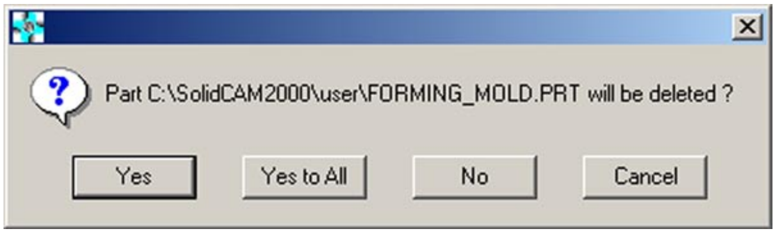

Part to delete
Cavity.PRT
cavity_msj.PRT
CONTOUR_PLATE.PRT
FORMING_MOLD.PRT
mill_turn_example.PRT
MOLD_INSERT.PRT
PRISMATIC_PLATE.PRT
RECOGNITION.PRT
TURNING.PRT
WCUT1.PRT
WCUT2.PRT
WCUT3.PRT
WCUT4.PRT
WCUT5.PRT
Delete
Exit

Look in: user

File name:

Files of type: CAM-Parts (*.prt)

1. Select the CAM-Part you want to delete. Use the Ctrl-key to select several CAM-Parts together.
2. Click on the **Delete** button.
3. Transfer the list of the CAM-Parts you want to delete to the right field by pressing the Left to Right arrow.
4. A **confirmation message** will be displayed:

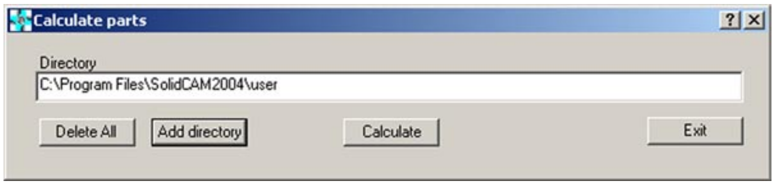


Part C:\SolidCAM2000\user\FORMING_MOLD.PRT will be deleted ?

5. Choose **Yes to all** to delete all selected CAM-Parts.

Calculate CAM-Parts

Calculates tool paths of all CAM-parts in the specified folder. The **Calculate CAM-Parts dialog** will be displayed.



Directory: C:\Program Files\SolidCAM2004\user

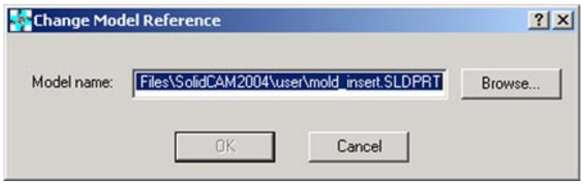
1. Click on the **Add Directory** button.
2. Choose the folder where the CAM-Part is situated via the standard Windows-browser.
3. Click on the **Calculate** button to start the operation.

You can also use the **Delete All** button to remove all the CAM-Parts from the specified folder.

Change Model Reference

SolidCAM enables you to change the reference to original SolidWorks model. This operation enables you to substitute the **DesignModel** component of the CAM-Part assembly with some SolidWorks model.

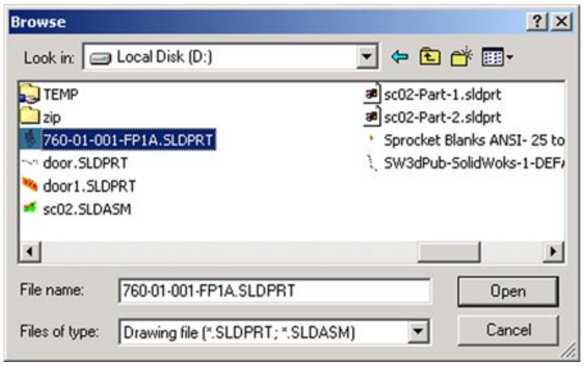
1. Choose the **Change Model Reference** command in the menu. The **Change Model Reference dialog** will be displayed.


Change Model Reference
? X
Browse...
OK
Cancel

Model name: Files\SolidCAM2004\user\mold_insert.SLDPRT

The **OK** button is disabled until the data in the dialog is changed.

2. Type the model name in the in the Model Name editbox or choose the model with the **Browse** button. The **Browse** button displays the browser dialog that enables you to choose the SolidWorks model.

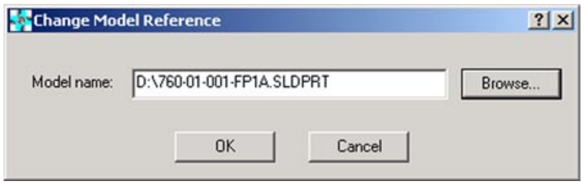

Browse
? X
TEMP
zip
760-01-001-FP1A.SLDPRT
door.SLDPRT
door1.SLDPRT
sc02.SLDASM
sc02-Part-1.sldprt
sc02-Part-2.sldprt
Sprocket Blanks ANSI- 25 to
SW3dPub-SolidWoks-1-DEF
Open
Cancel

Look in: Local Disk (D:)

File name: 760-01-001-FP1A.SLDPRT

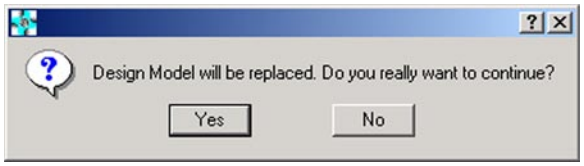
Files of type: Drawing file (*.SLDPRT; *.SLDASM)

When the data in the **Change Model Reference dialog** is changed, the **OK** button will be enabled.


Change Model Reference
? X
Browse...
OK
Cancel

Model name: D:\760-01-001-FP1A.SLDPRT

3. Click on the **OK** button. The following message will be displayed.

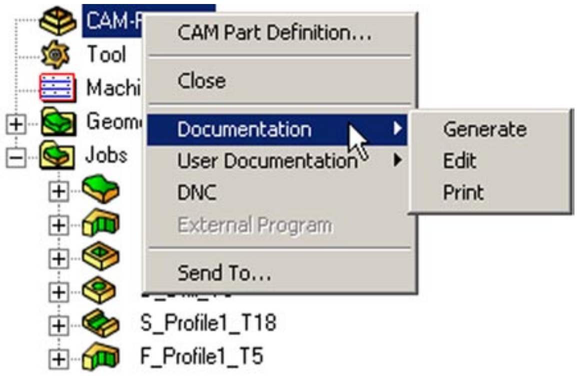


Design Model will be replaced. Do you really want to continue?

When the message is confirmed, the CAM-Part will be closed after which the **CAM-Part Reference** will be updated, and then the updated CAM-Part will be loaded again.

2.11 CAM-Part Documentation

Documentation enables you to document your tool list, job sequence etc.


CAM-P
Tool
Machi
Geom
Jobs
S_Profile1_T18
F_Profile1_T5
CAM Part Definition...
Close
Documentation
User Documentation
DNC
External Program
Send To...
Generate
Edit
Print

- **Generate**

  Pressing on **Generate** will create the document.

- **Edit**

  This option enables you to edit the document after it has been generated.

- **Print**

  This option enables you to print the document that has been generated.

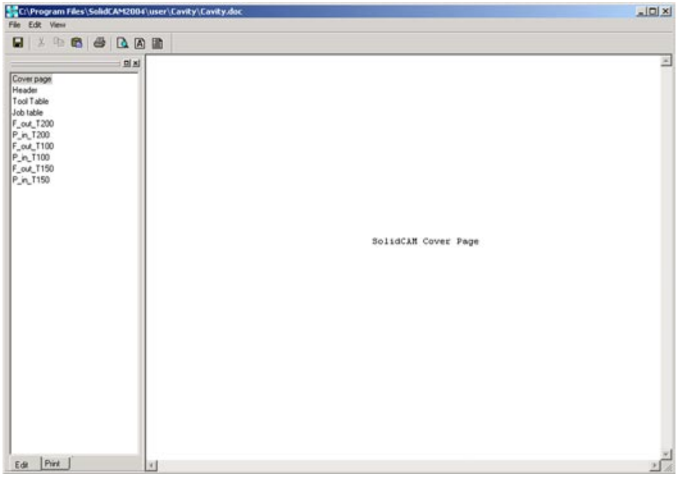
2.11.1 Documentation Editor

With the documentation editor you can browse the generated CAM-Part documentation. The tool bar and application menu on the top of the documentation editor works the same as in any typical windows application.

The left panel enables you to switch pages and the right page enables you to view and edit each page.

- **Cover page**

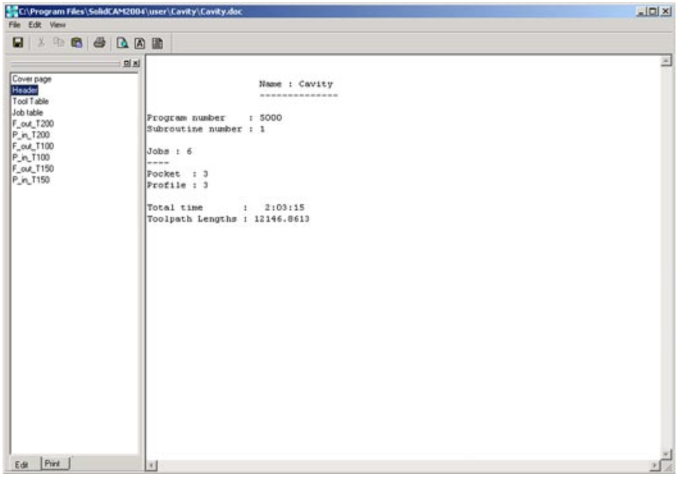
  Shows contents of Cover page file defined in SolidCAM Settings.


C:\Program Files\SolidCAM2004\user\Cavity\Cavity.doc
File
Edit
View
Cover page
Header
Tool Table
Job table
F_out_T200
P_in_T200
F_out_T100
P_in_T100
F_out_T150
P_in_T150
Edit
Print

SolidCAM Cover Page

- **Header page**

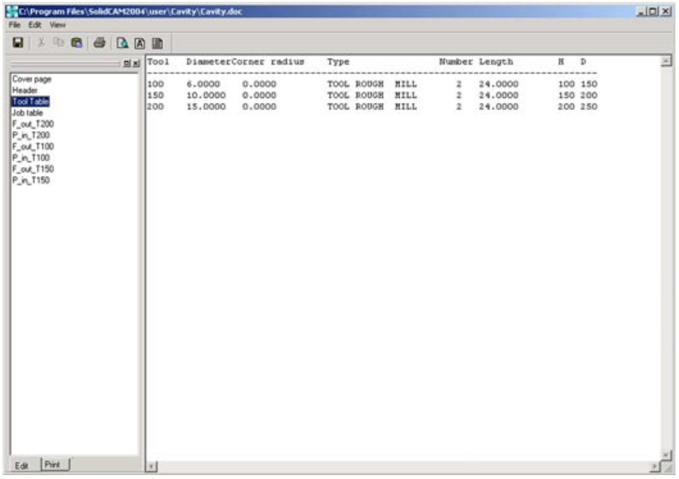
  The **Header page** contains general information about the CAM-Part. It will give you the CAM-Part name, program number, the amount of jobs and the type of jobs. To edit, simply click the mouse in the right field and make any changes or additions.


C:\Program Files\SolidCAM2004\user\Cavity\Cavity.doc
File
Edit
View
Cover page
Header
Tool Table
Job table
F_out_T200
P_in_T200
F_out_T100
P_in_T100
F_out_T150
P_in_T150
Edit
Print

```
                    Name : Cavity
                    --------------

Program number     : 5000
Subroutine number : 1

Jobs : 6
----
Pocket   : 3
Profile : 3

Total time        :   2:03:15
Toolpath Lengths : 12146.8613
```

- **Tool Table page**

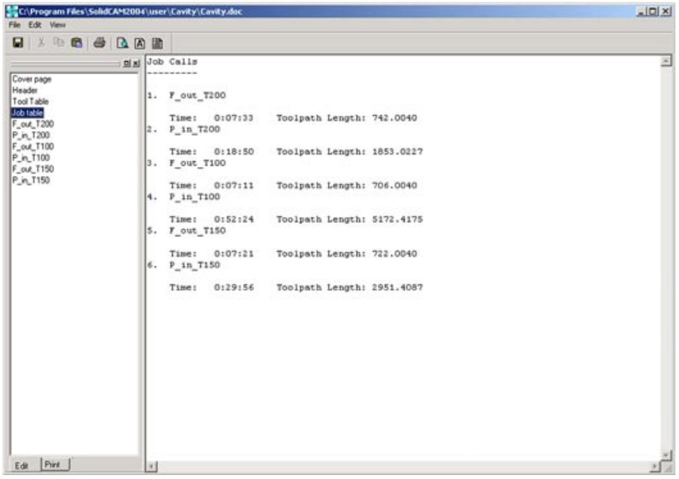
The **Tool Table page** contains all the relevant information about the tools used in the CAM-Part. It will list the tool number, tool diameter, corner radius, tool type, number of teeth and any other information that is relevant to each tool. To edit, simply click the mouse in the right field and make any changes or additions.


C:\Program Files\SolidCAM2004\user\Cavity\Cavity.doc
File
Edit
View
Cover page
Header
Tool Table
Job table
F_out_T200
P_in_T200
F_out_T100
P_in_T100
F_out_T150
P_in_T150
Edit
Print

```
Tool    DiameterCorner radius     Type                  Number Length        H   D
-----------------------------------------------------------------------------------
100     6.0000     0.0000         TOOL ROUGH  MILL         2   24.0000       100 150
150     10.0000    0.0000         TOOL ROUGH  MILL         2   24.0000       150 200
200     15.0000    0.0000         TOOL ROUGH  MILL         2   24.0000       200 250
```

- **Jobs table page**

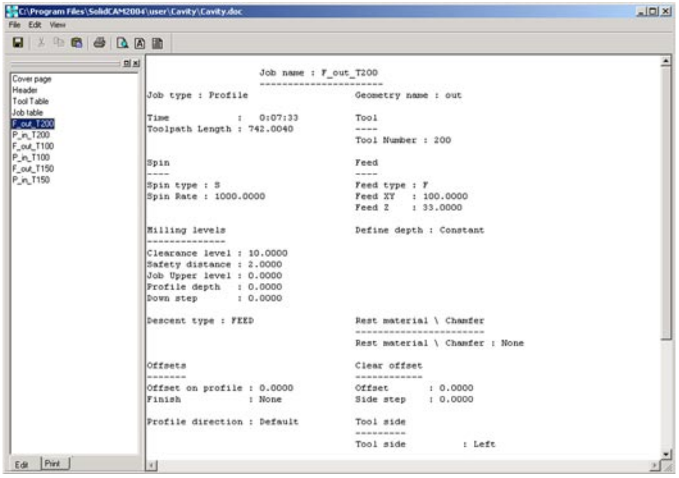
The **Job table page** contains the list of all the jobs in the CAM-Part.


C:\Program Files\SolidCAM2004\user\Cavity\Cavity.doc
File
Edit
View
Cover page
Header
Tool Table
Job table
F_out_T200
P_in_T200
F_out_T100
P_in_T100
F_out_T150
P_in_T150
Edit
Print

```
Job Calls
---------

1.  F_out_T200

    Time:   0:07:33    Toolpath Length: 742.0040
2.  P_in_T200

    Time:   0:18:50    Toolpath Length: 1853.0227
3.  F_out_T100

    Time:   0:07:11    Toolpath Length: 706.0040
4.  P_in_T100

    Time:   0:52:24    Toolpath Length: 5172.4175
5.  F_out_T150

    Time:   0:07:21    Toolpath Length: 722.0040
6.  P_in_T150

    Time:   0:29:56    Toolpath Length: 2951.4087
```

- **Job pages**

Under the **Job table page**, there is a page for each job name. This page gives you a detailed list of all the fields you have chosen in the job and their values.


C:\Program Files\SolidCAM2004\user\Cavity\Cavity.doc
File
Edit
View
Cover page
Header
Tool Table
Job table
F_out_T200
P_in_T200
F_out_T100
P_in_T100
F_out_T150
P_in_T150
Edit
Print

```
                    Job name : F_out_T200
                    ----------------------
Job type : Profile                  Geometry name : out

Time            :   0:07:33         Tool
Toolpath Length : 742.0040          ----
                                    Tool Number : 200


Spin                                Feed
----                                ----
Spin type : S                       Feed type : F
Spin Rate : 1000.0000               Feed XY   : 100.0000
                                    Feed Z    : 33.0000

Milling levels                      Define depth : Constant
--------------
Clearance level : 10.0000
Safety distance : 2.0000
Job Upper level : 0.0000
Profile depth   : 0.0000
Down step       : 0.0000

Descent type : FEED                 Rest material \ Chamfer
                                    -----------------------
                                    Rest material \ Chamfer : None


Offsets                             Clear offset
-------                             ------------
Offset on profile : 0.0000          Offset       : 0.0000
Finish            : None            Side step    : 0.0000

Profile direction : Default         Tool side
                                    ---------
                                    Tool side         : Left
```

Tools

3

The tool commands can be used to create and manage tools and tool tables in SolidCAM. You can shorten your programming time if you save the tools you use most often in customizable tool tables.

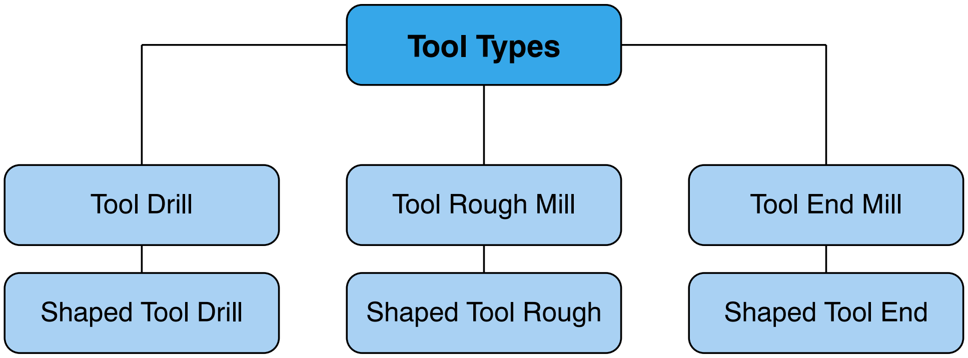
The tool tables can be based on machine tooling, on materials or any other personal criteria. When you define your machining operations, you can then load the tool and all its parameters, feed and spin values are automatically copied to the job.

❒ Working with Part Tool Table
❒ Working with Current Tool Table
❒ Managing Tool Tables
❒ Defining tools
❒ Tool types
❒ Shaped tools
❒ Tool holding system

3.1 User-defined Tool Types

For easier identification and classification of tools, you can define your own tool types. SolidCAM offers three default tool types, namely:

- **Tool Rough Mill** for rough cutters.
- **Tool End Mill** for finish cutters.
- **Drill** for drills, bores, reams, threading tools etc...
- **Shaped Tool Drill** for shaped drills, reams etc...
- **Shaped Tool Rough** for shaped rough cutters.
- **Shaped Tool End Mill** for shaped finish cutters.

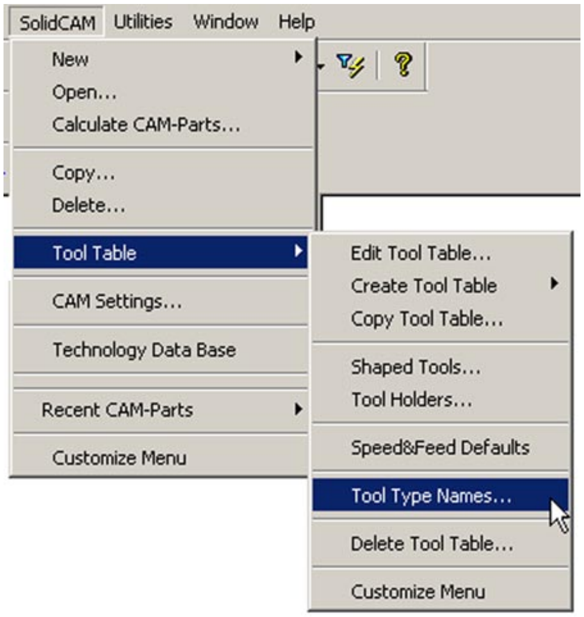

Tool Types
Tool Drill
Tool Rough Mill
Tool End Mill
Shaped Tool Drill
Shaped Tool Rough
Shaped Tool End

User-defined tool types can also be used to define new tools and to search tool tables.

3.1.1 Add new Tool Types

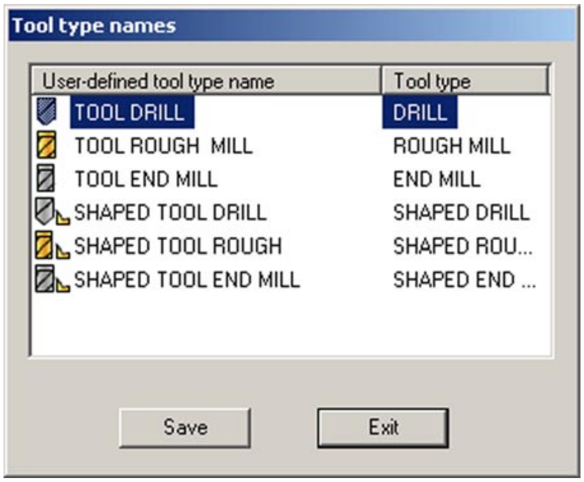
Based on one of the three default tool types, you can create user-defined tool types.

1. Choose the **SolidCAM** item from the SolidWorks main menu. Choose the **Tool type names…** item from the Tool Table sub-menu.


SolidCAM
Utilities
Window
Help
New
Open…
Calculate CAM-Parts…
Copy…
Delete…
Tool Table
CAM Settings…
Technology Data Base
Recent CAM-Parts
Customize Menu
Edit Tool Table…
Create Tool Table
Copy Tool Table…
Shaped Tools…
Tool Holders…
Speed&Feed Defaults
Tool Type Names…
Delete Tool Table…
Customize Menu

The **Tool types names dialog** will be displayed. This dialog shows the user-defined tool type names and their actual SolidCAM default internal tool types.

2. In the **Tool types names dialog**, move the cursor on an existing tool type name, e.g. TOOL END MILL and press the **right** mouse button.

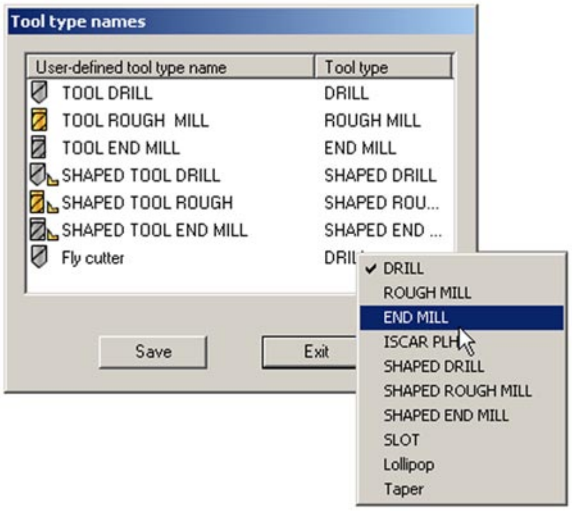

Tool type names
Save
Exit

User-defined tool type name	Tool type
TOOL DRILL	DRILL
TOOL ROUGH MILL	ROUGH MILL
TOOL END MILL	END MILL
SHAPED TOOL DRILL	SHAPED DRILL
SHAPED TOOL ROUGH	SHAPED ROU...
SHAPED TOOL END MILL	SHAPED END ...

3. Select **Add** from the menu.

4. A new entry will be created. Enter the name of the user-defined tool-type.

5. By default, the new entry will be of tool type **DRILL**. With the **right** mouse button, click on the **DRILL** field and select the relevant tool type from the list.


Tool type names
DRILL
ROUGH MILL
END MILL
ISCAR PLH
SHAPED DRILL
SHAPED ROUGH MILL
SHAPED END MILL
SLOT
Lollipop
Taper
Save
Exit

User-defined tool type name	Tool type
TOOL DRILL	DRILL
TOOL ROUGH MILL	ROUGH MILL
TOOL END MILL	END MILL
SHAPED TOOL DRILL	SHAPED DRILL
SHAPED TOOL ROUGH	SHAPED ROU...
SHAPED TOOL END MILL	SHAPED END ...
Fly cutter	DRIL

6. After you have completed your changes in the tool type list, click on the **Save** button to confirm your changes.

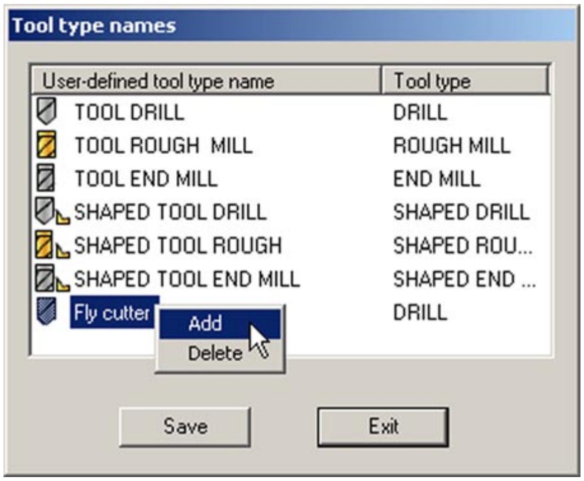
Click the tool type name twice to **re-name** existing entries.

3.1.2 Delete Tool Types

You can delete user-defined tool types from the list.

1. In the **Names List** dialog, move the cursor on an existing tool type name, e.g. TOOL END MILL and press the **right** mouse button.

2. Choose the **Delete** item from the menu and confirm the displayed confirmation message with **OK**.


Tool type names
Add
Delete
Save
Exit

User-defined tool type name	Tool type
TOOL DRILL	DRILL
TOOL ROUGH MILL	ROUGH MILL
TOOL END MILL	END MILL
SHAPED TOOL DRILL	SHAPED DRILL
SHAPED TOOL ROUGH	SHAPED ROU...
SHAPED TOOL END MILL	SHAPED END ...
Fly cutter	DRILL

3.2 Tool Tables

The **tool tables** can be based on machine tooling, on materials or any other personal criteria. When you define your machining operations, you can then load the tool and all its parameters, feed and spin values are automatically copied to the job.

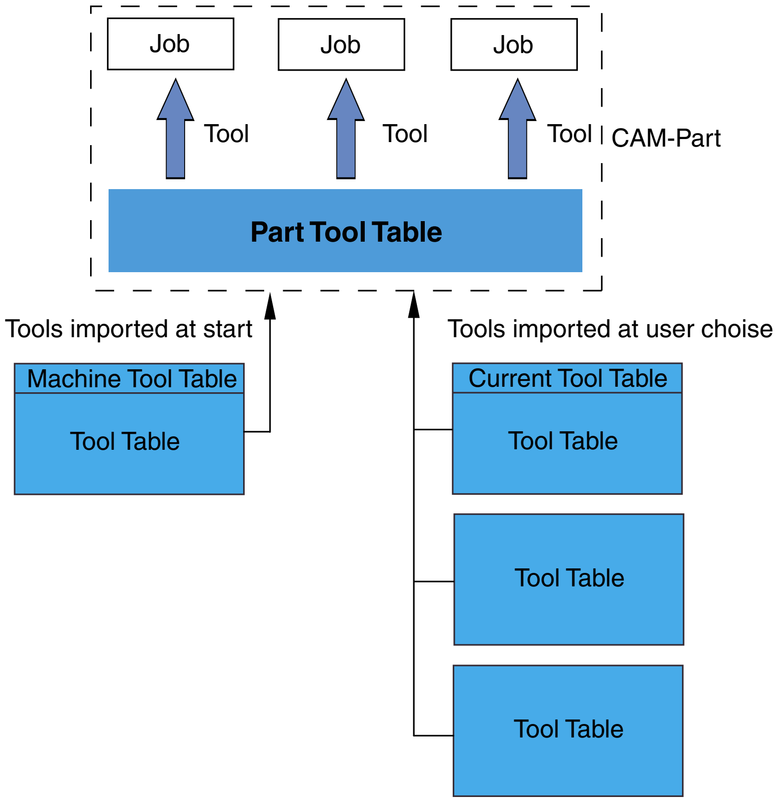
The **Part Tool Table** is the tool table that contains all the tools that are available for use within a specific CAM-Part. The **Part Tool Table** is stored within the CAM-Part.

When you start a new CAM-Part and you want to automatically copy the contents of a specific tool table into the **Part Tool Table** of this CAM-Part, specify its name in the MAC-file of the CNC machine; this file is called **Machine Tool Table**.

The **Machine Tool table** is defined in the MAC-file by the following parameter: **tool_table_name = name**; where **name** (without extension) is the name of the Tool Table located in the SolidCAM Tool tables folder defined in the **SolidCAM settings** (see the topic **10.2.2**).

Generally, the **Machine Tool Table** contains the tools situated in the Tool Magazine of the specified CNC Machine.

SolidCAM enables you to define any number of **Tool Tables**. Each tool from such Tool Tables can be imported into the **Part Tool Table**. SolidCAM enables you to define one Tool Table that has a top priority in the list for import; this is the **Current Tool Table**.

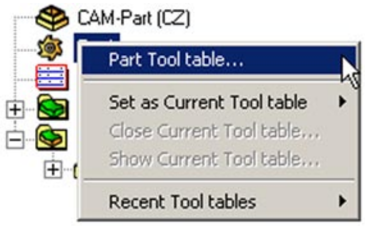

Job
Job
Job
Tool
Tool
Tool
CAM-Part
Part Tool Table
Tools imported at start
Tools imported at user choise
Machine Tool Table
Tool Table
Current Tool Table
Tool Table
Tool Table
Tool Table

3.3 Working with Part Tool Table

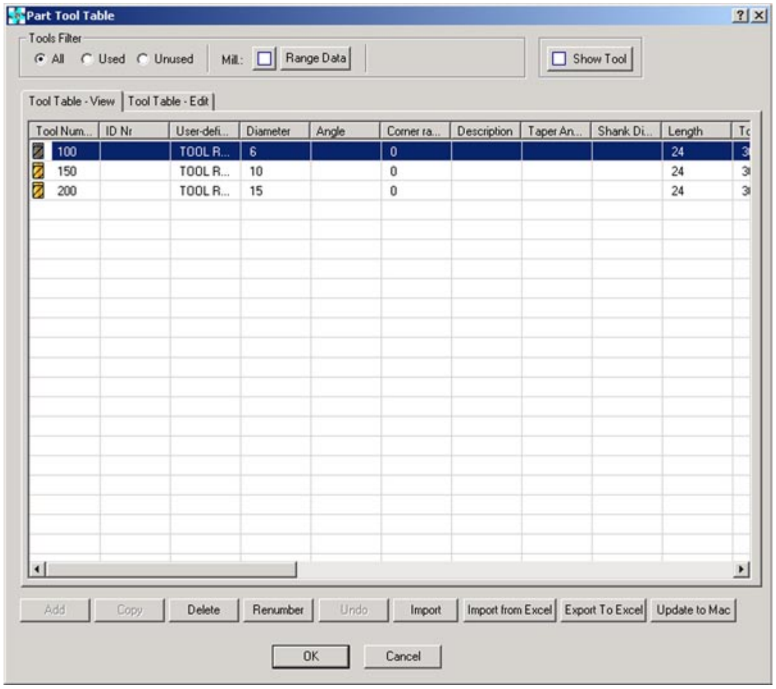
The **Part Tool Table** is the tool table that contains all the tools that are available for use within a specific CAM-Part. The **Part Tool Table** is stored within the CAM-Part.

SolidCAM enables you to display the **Part Tool Table** dialog (see the topic **3.6**). This dialog enables you to manage the tools contained in the Part Tool Table.

1. Choose the **Part Tool table** option from the right-click menu on the Tool header in the CAM Manager.


CAM-Part (CZ)
Part Tool table...
Set as Current Tool table
Close Current Tool table...
Show Current Tool table...
Recent Tool tables

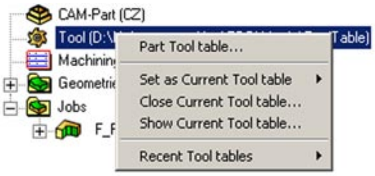
2. The **Part Tool Table** dialog will be displayed.


Part Tool Table
Tools Filter
● All
○ Used
○ Unused
Mill:
☐
Range Data
☐ Show Tool
Tool Table - View
Tool Table - Edit
Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
Update to Mac
OK
Cancel

Tool Num...	ID Nr	User-defi...	Diameter	Angle	Corner ra...	Description	Taper An...	Shank Di...	Length	Tc
100		TOOL R...	6		0				24	3
150		TOOL R...	10		0				24	3
200		TOOL R...	15		0				24	3

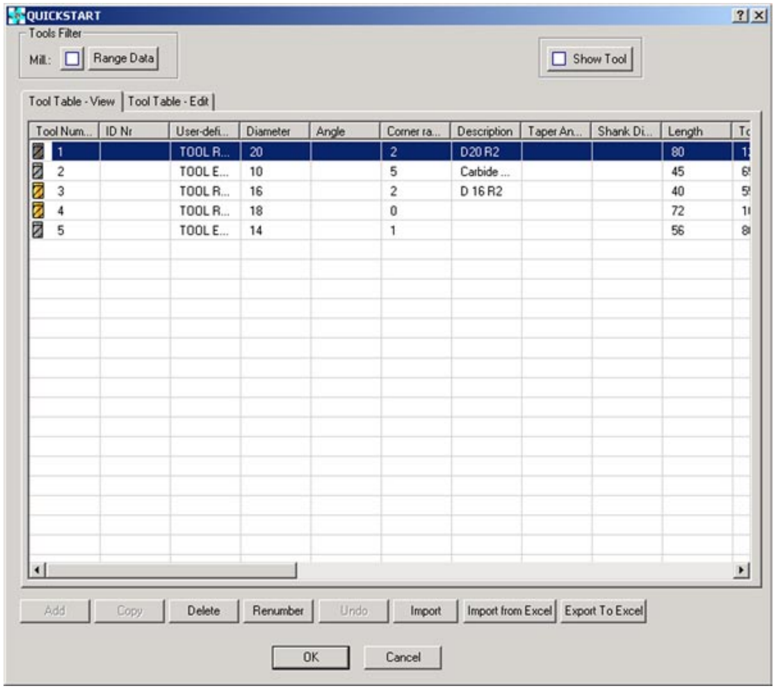
3.4 Working with Current Tool Table

SolidCAM enables you to control your **Current Tool Tables** with the right-click menu available on the Tool header in the SolidCAM Manager.


CAM-Part (CZ)
Tool (D:\ Table)
Machinin
Geometri
Jobs
F_F
Part Tool table...
Set as Current Tool table
Close Current Tool table...
Show Current Tool table...
Recent Tool tables

• Set as Current Tool Table

This option enables you to set the **Current Tool Table**. You have to specify the type of the Current Tool Table (Milling, Turning or Milling&Turning) and choose the necessary Tool Table with the browser-styled dialog. The **Current Tool table** will be selected and displayed in the **Tool Table dialog** (see the topic **3.6**).


QUICKSTART
Tools Filter
Mill:
Range Data
Show Tool
Tool Table - View
Tool Table - Edit
Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
OK
Cancel

Tool Num...	ID Nr	User-defi...	Diameter	Angle	Corner ra...	Description	Taper An...	Shank Di...	Length	Tc
1		TOOL R...	20		2	D20 R2			80	1
2		TOOL E...	10		5	Carbide ...			45	6
3		TOOL R...	16		2	D 16 R2			40	5
4		TOOL R...	18		0				72	1
5		TOOL E...	14		1				56	8

• Close Current Tool Table

This option enables you to close the **Current Tool Table**.

This option is active only when the **Current Tool table** is already defined.

- **Show Current Tool Table**

  This option enables you to display the content of the **Current Tool tables**. The **Tool Table dialog** enables you to manage the tools in the **Current Tool Table** and edit them.

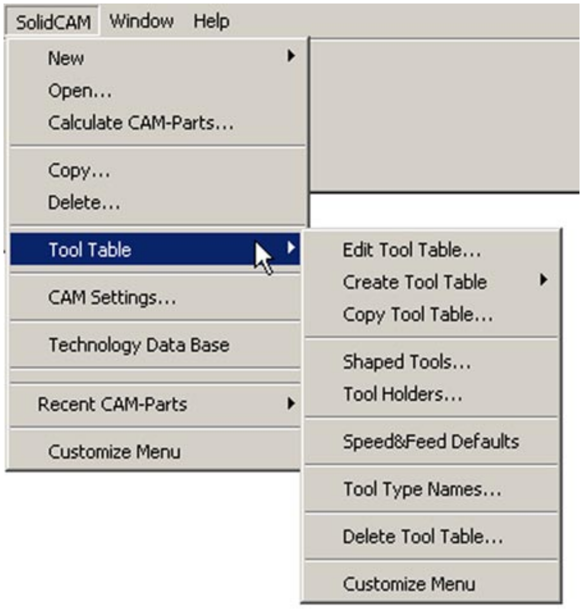
  This option is active only when the **Current Tool table** is already defined.

- **Recent Tool Tables**

  This option displays a list of the recently loaded **Current Tool Tables**. You can open the file by simply clicking on its name. The **Tool Table dialog** will be displayed.

3.5 Managing tool tables

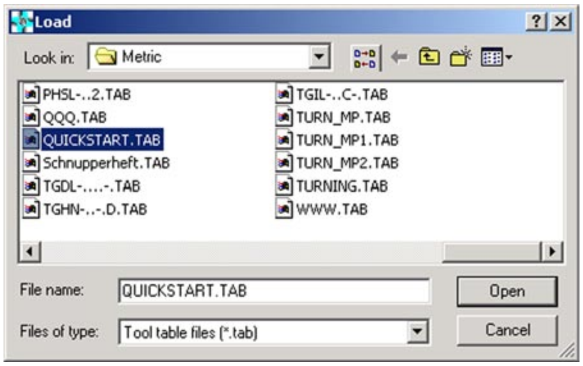
SolidCAM enables you to manage your **Tool Tables** with the **Tool Table menu** available from the SolidCAM main menu.


SolidCAM
Window
Help
New
Open...
Calculate CAM-Parts...
Copy...
Delete...
Tool Table
CAM Settings...
Technology Data Base
Recent CAM-Parts
Customize Menu
Edit Tool Table...
Create Tool Table
Copy Tool Table...
Shaped Tools...
Tool Holders...
Speed&Feed Defaults
Tool Type Names...
Delete Tool Table...
Customize Menu

3.5.1 Edit Tool table

This option enables you to display the content of the specific **Tool Table**.

When you choose this option, the **Load** dialog will be displayed.


Load
Look in:
Metric
PHSL-..2.TAB
QQQ.TAB
QUICKSTART.TAB
Schnupperheft.TAB
TGDL-....-.TAB
TGHN-..-.D.TAB
TGIL-..C-.TAB
TURN_MP.TAB
TURN_MP1.TAB
TURN_MP2.TAB
TURNING.TAB
WWW.TAB
Open
Cancel

File name: QUICKSTART.TAB

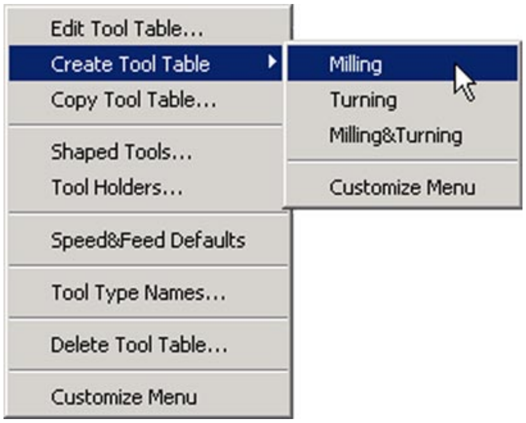
Files of type: Tool table files (*.tab)

1. Select the tool table you want to load.
2. Double click on the tool table or select it and click on the **Open** button. The **Tool Table dialog** will be displayed. This dialog enables you to manage the tools in the chosen Tool Table and edit them.

3.5.2 Create Tool table

This option enables you to define a new **Tool Table**.

1. Choose the Tool Table type (Milling, Turning or Milling&Turning) from the submenu.


Edit Tool Table...
Create Tool Table
Copy Tool Table...
Shaped Tools...
Tool Holders...
Speed&Feed Defaults
Tool Type Names...
Delete Tool Table...
Customize Menu
Milling
Turning
Milling&Turning
Customize Menu

2. Define the Tool table name

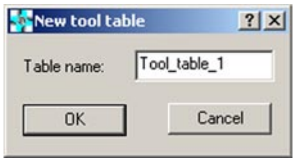

New tool table
OK
Cancel

Table name: Tool_table_1

3. Confirm the Tool table creation with the **OK** button. The Tool Table will be created. The **Tool Table dialog** will be displayed enabling you to add tools to the new Tool Table.

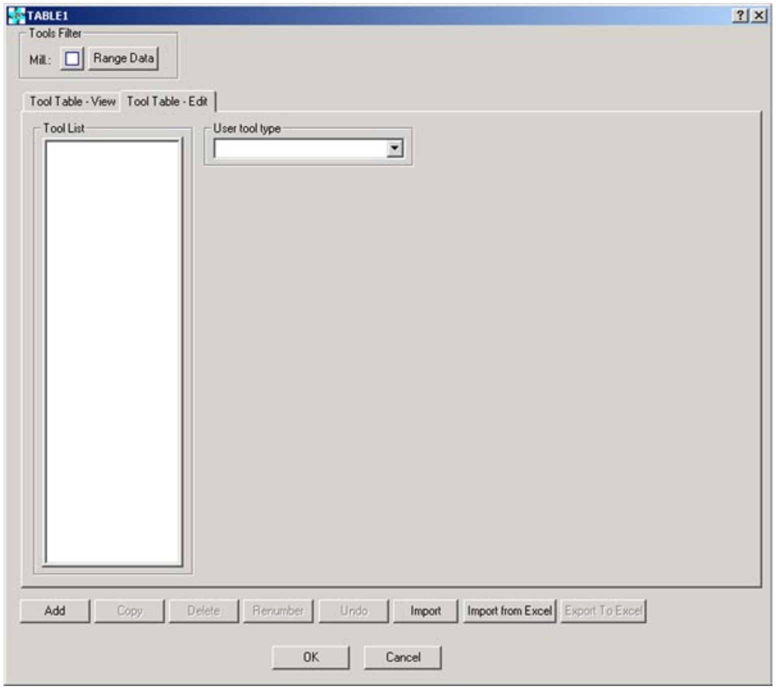
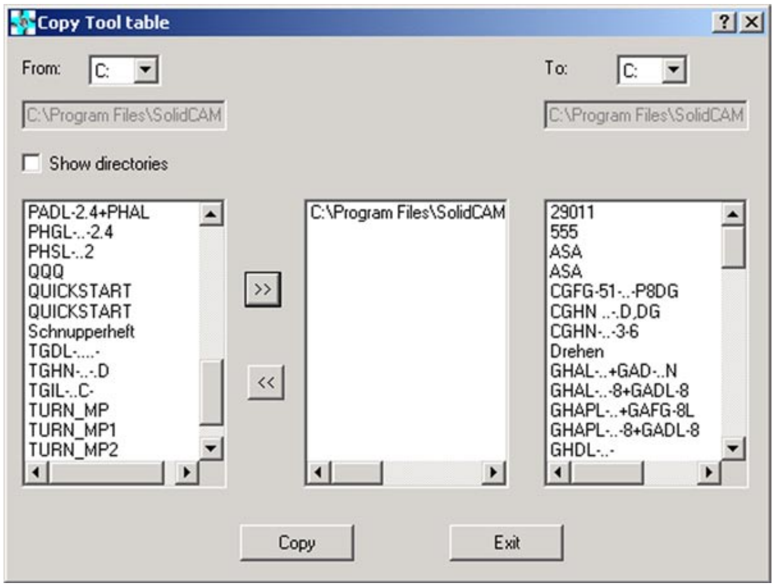


TABLE1
Tools Filter
Mill:
Range Data
Tool Table - View
Tool Table - Edit
Tool List
User tool type
Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
OK
Cancel

3.5.3 Copy tool table

Creates an identical copy of a tool table in another folder or drive for backup purposes or for making modifications without changing the original. The **Copy tool table** dialog box will be displayed.



From: C:

C:\Program Files\SolidCAM

☐ Show directories

PADL-2.4+PHAL
PHGL-..-2.4
PHSL-..2
QQQ
QUICKSTART
QUICKSTART
Schnupperheft
TGDL-....-
TGHN-..-.D
TGIL-..C-
TURN_MP
TURN_MP1
TURN_MP2

C:\Program Files\SolidCAM

To: C:

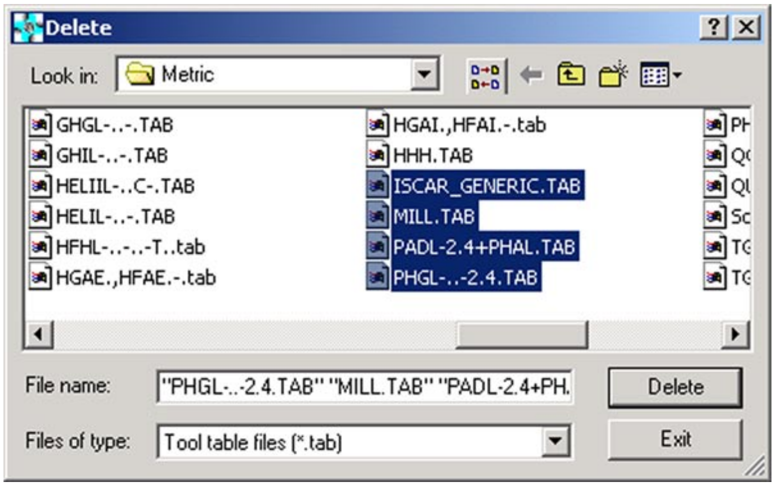
C:\Program Files\SolidCAM

29011
555
ASA
ASA
CGFG-51-..-P8DG
CGHN ..-.D,DG
CGHN-..-3-6
Drehen
GHAL-..+GAD-..N
GHAL-..-8+GADL-8
GHAPL-..+GAFG-8L
GHAPL-..-8+GADL-8
GHDL-..-

1. In the left window, choose the drive and directory from which you want to copy the original tool table.
2. Select the tool table you want to copy.
3. Transfer the selection of tool tables you want to copy to the middle window by pressing the Left to Right arrow.
4. Choose the drive and the directory you want to copy in on the right window.
5. Press the **Copy** button to execute the copy command.

3.5.4 Delete tool table

This field enables you to delete tool tables. The **Delete Tool tables** dialog box will be displayed.


Delete
GHGL-..-.TAB
GHIL-..-.TAB
HELIIL-..C-.TAB
HELIL-..-.TAB
HFHL-..-..-T..tab
HGAE.,HFAE.-.tab
HGAI.,HFAI.-.tab
HHH.TAB
ISCAR_GENERIC.TAB
MILL.TAB
PADL-2.4+PHAL.TAB
PHGL-..-2.4.TAB
Delete
Exit

Look in: Metric

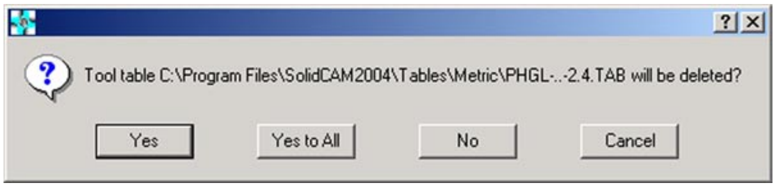
File name: "PHGL-..-2.4.TAB" "MILL.TAB" "PADL-2.4+PH.

Files of type: Tool table files (*.tab)

1. Select the tool tables you want to delete. Use the Ctrl-key to select several tool tables together.

2. Click on the **Delete** button.

3. A confirmation message will be displayed:

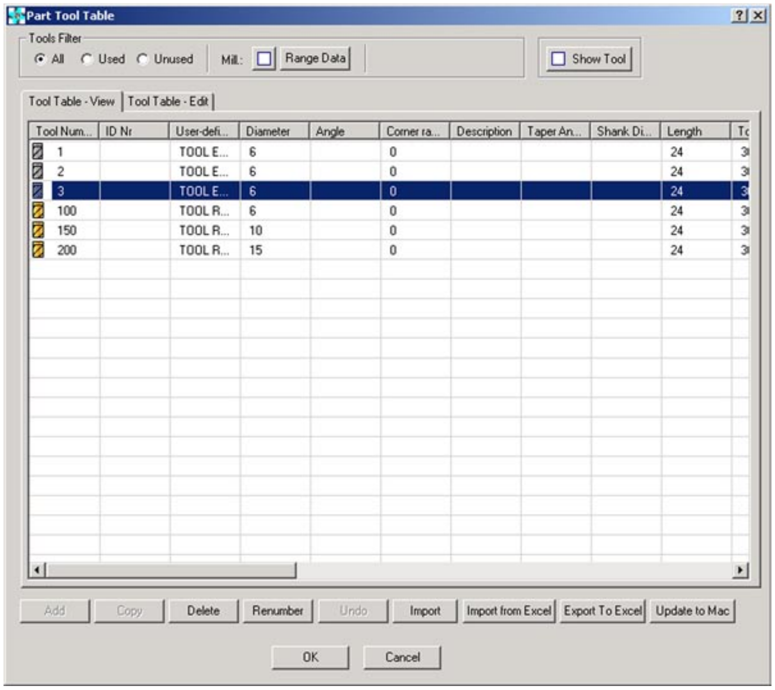

Yes
Yes to All
No
Cancel

Tool table C:\Program Files\SolidCAM2004\Tables\Metric\PHGL-..-2.4.TAB will be deleted?

4. Choose **Yes** or **Yes to all** to delete all selected tool tables.

3.6 Tool Table dialog

The **Tool table** dialog contains structured information about all tools included in the specified Tool Table. This dialog enables you to add new tools to the Tool Table, remove tools from the tool Table, edit the tool definition etc.



Part Tool Table
Tools Filter
Range Data
Tool Table - View
Tool Table - Edit
Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
Update to Mac
OK
Cancel

Tools Filter: ● All ○ Used ○ Unused Mill: ☐

☐ Show Tool

Tool Num...	ID Nr	User-defi...	Diameter	Angle	Corner ra...	Description	Taper An...	Shank Di...	Length	Tc
1		TOOL E...	6		0				24	3
2		TOOL E...	6		0				24	3
3		TOOL E...	6		0				24	3
100		TOOL R...	6		0				24	3
150		TOOL R...	10		0				24	3
200		TOOL R...	15		0				24	3

The dialog contains two pages:

- **Tool table - View**

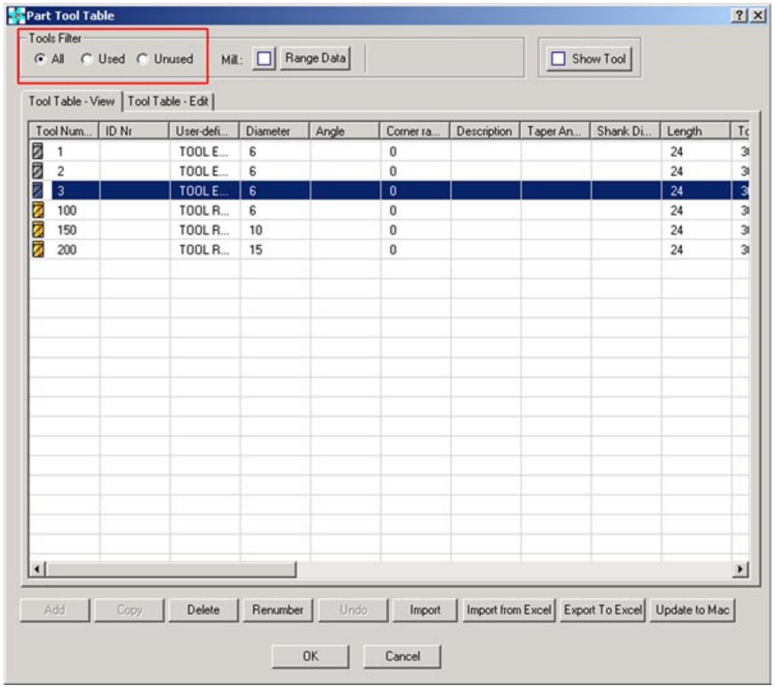
  This page displays the tools of the Tool Table in table format.

- **Tool table - Edit**

  This page enables the addition of new tools or editing of existing tools in the Tool Table.

3.6.1 Tools Filter

SolidCAM enables you to filter tools that will be displayed in the Tool Table dialog.


Part Tool Table
Tools Filter
● All ○ Used ○ Unused
Mill: ☐
Range Data
☐ Show Tool
Tool Table - View
Tool Table - Edit
Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
Update to Mac
OK
Cancel

Tool Num...	ID Nr	User-defi...	Diameter	Angle	Corner ra...	Description	Taper An...	Shank Di...	Length	Tc
1		TOOL E...	6		0				24	3
2		TOOL E...	6		0				24	3
3		TOOL E...	6		0				24	3
100		TOOL R...	6		0				24	3
150		TOOL R...	10		0				24	3
200		TOOL R...	15		0				24	3

- **All**

  All the tools will be displayed.

- **Used**

  Only the tools used by Jobs of the current CAM-Part will be displayed.

- **Unused**

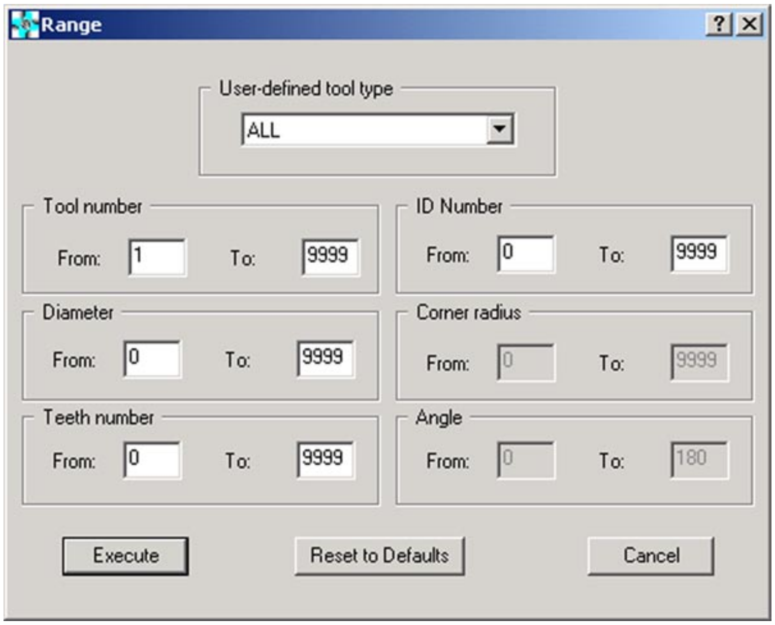
  Only the tools unused by Jobs of the current CAM-Part will be displayed.

The filter is available only for the Part Tool table.

3.6.2 Tool Range

If you work with large tool tables, you can use this option to reduce the number of tools displayed. You can also search the tool table for tools that match your criteria.

In the **Tool Range dialog**, enter the required properties of the tool. Enter only values that are different; leave the other fields unchanged.



User-defined tool type: ALL

Tool number From: 1 To: 9999

ID Number From: 0 To: 9999

Diameter From: 0 To: 9999

Corner radius From: 0 To: 9999

Teeth number From: 0 To: 9999

Angle From: 0 To: 180

Sort by enables you to list the search results according to:

- **Tool number**. Lists results according to the tool numbers.
- **Sequence number**. Lists results according to the sequence in which the tools were defined.

User-defined tool type allows you to choose the **type of tool** you would like to have listed in the tool table.

ID Number allows you to choose the range of the **ID Number** of the tools you would like to have listed in the tool table. If, for example, you would like to list the tools that have an **ID Number** from **10** to **100**, enter in the **From** field **10** and in the **To** field **100**.

Diameter allows you to choose the range using the diameter of the tools you would like to have listed in the tool table. If, for example, you would like to list the tools that have a diameter from **1** to **16**, enter in the **From** field **1** and in the **To** field **16**.

Teeth number allows you to choose the range using the number of teeth of the tools you would like to have listed in the tool table. If, for example, you would like to list the tools that have from **1** to **4**, enter in the **From** field **1** and in the **To** field **4**.

Tool number allows you to choose the range of the tool number of the tools you would like to have listed in the tool table. If, for example, you would like to list the tools that have the tool number from **1** to **30**, enter in the **From** field **1** and in the **To** field **30**.

Sequence number allows you to choose the range of the tool sequence number of the tools you would like to have listed in the tool table. If, for example, you would like to list the tools that have the tool sequence number from **1** to **30**, enter in the **From** field **1** and in the **To** field **30**.

Corner radius allows you to choose the range using the corner radius of the tools you would like to have listed in the tool table. If, for example, you would like to list the tools that have a corner radius from **0** to **4**, enter in the **From** field **0** and in the **To** field **4**.

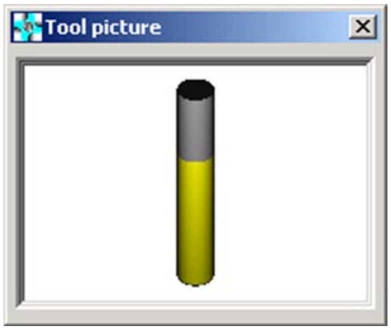
Angle allows you to choose the range using the angle of the tools you would like to have listed in the tool table. If, for example, you would like to list the tools that have an angle from **90** to **180**, enter in the **From** field **90** and in the **To** field **180**.

Execute. Click on the **Execute** button and the tool table will be searched for the tools that match your requirements. SolidCAM will return to the tool table and display the result of the range search.

The Reset to defaults button returns you to the original program settings of this dialog.

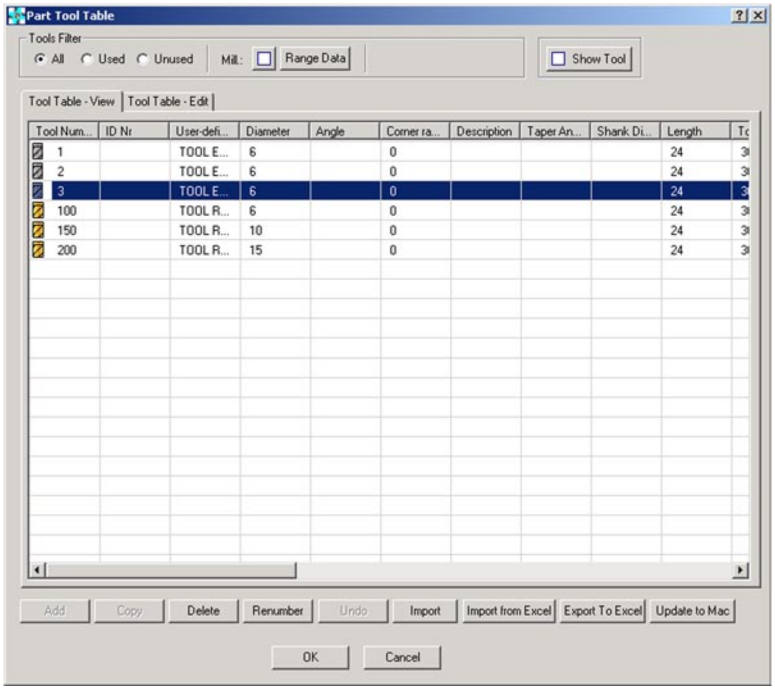
3.6.3 Show Tool

This option enables you to display the selected tool.



3.6.4 Tool table - View

This page displays the tools data in table format. The table contains the following columns:


Part Tool Table
Tools Filter
● All ○ Used ○ Unused
Mill: ☐
Range Data
☐ Show Tool
Tool Table - View
Tool Table - Edit
Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
Update to Mac
OK
Cancel

Tool Num...	ID Nr	User-defi...	Diameter	Angle	Corner ra...	Description	Taper An...	Shank Di...	Length	Tc
1		TOOL E...	6		0				24	3
2		TOOL E...	6		0				24	3
3		TOOL E...	6		0				24	3
100		TOOL R...	6		0				24	3
150		TOOL R...	10		0				24	3
200		TOOL R...	15		0				24	3

- **Tool Number**

  Displays the tool number in the Part Tool Table.

- **ID Number**

  Displays the ID number of the tool

- **Tool type name**

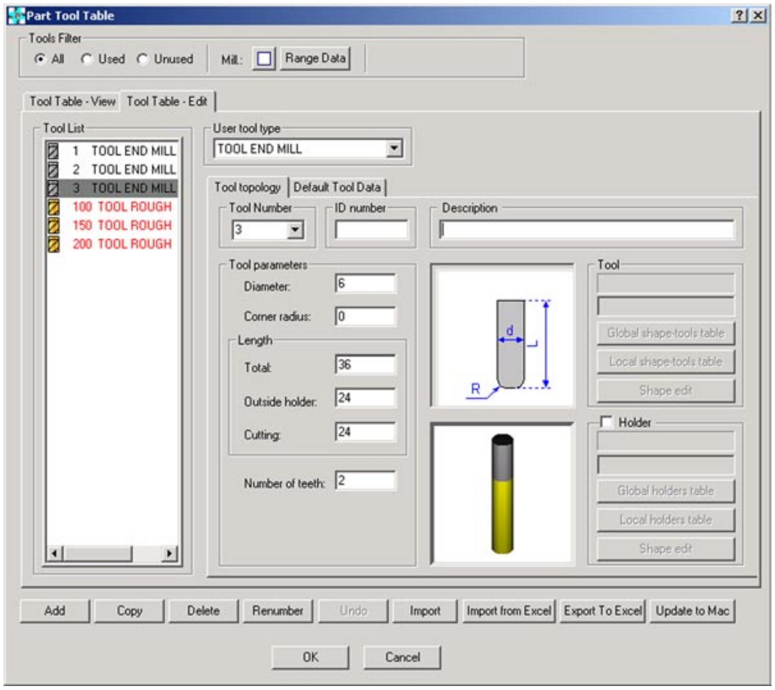
  Shows the tool type.

- **Description**

  Displays a description of the tool.

This page also displays the geometry parameters of the tool depending on the tool type.

3.6.4 Tool table - Edit



Tools Filter: ● All ○ Used ○ Unused

Tool List:
- 1 TOOL END MILL
- 2 TOOL END MILL
- 3 TOOL END MILL
- 100 TOOL ROUGH
- 150 TOOL ROUGH
- 200 TOOL ROUGH

User tool type: TOOL END MILL

Tool Number: 3

Tool parameters:
- Diameter: 6
- Corner radius: 0
- Length:
  - Total: 36
  - Outside holder: 24
  - Cutting: 24
- Number of teeth: 2

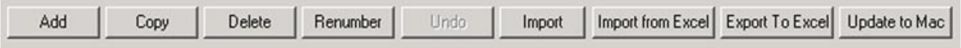
☐ Holder

Part Tool Table
Mill:
Range Data
Tool Table - View
Tool Table - Edit
Tool topology
Default Tool Data
ID number
Description
d
L
R
Tool
Global shape-tools table
Local shape-tools table
Shape edit
Global holders table
Local holders table
Shape edit
Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
Update to Mac
OK
Cancel

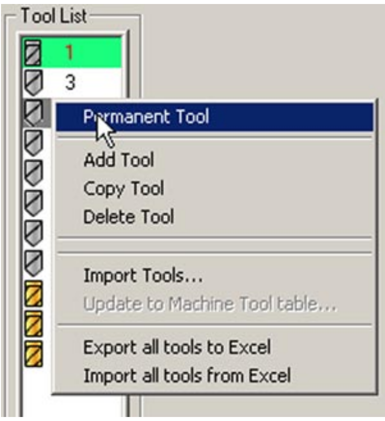
This page enables the addition of new tools or editing of the existing tools in the Tool Table.

3.7 Managing tools

SolidCAM enables you to manage the tools in the tool table with the buttons at the bottom of the dialog.


Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
Update to Mac

All the operations are also available with the right-click menu.


Tool List
1
3
Permanent Tool
Add Tool
Copy Tool
Delete Tool
Import Tools...
Update to Machine Tool table...
Export all tools to Excel
Import all tools from Excel

The following options are available:

Permanent Tool

If a particular tool that is being used is always in the same magazine position in the machine, the tool can be marked as a **Permanent Tool**. If you use the Tool Renumbering function, a tool that is marked as a **Permanent Tool** will not be renumbered.

Add Tool

This option enables you to add a new tool.

Copy Tool

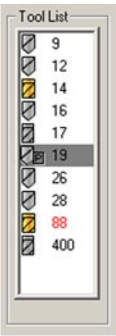
This option enables you to create a copy of an existing tool.

Delete Tool

This option enables you to delete the selected tool.

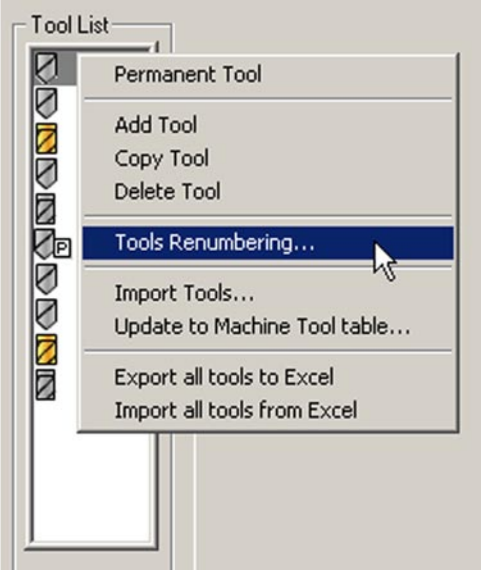
Tools Renumbering

This option enables you to renumber the tools.

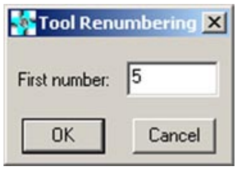

Tool List
9
12
14
16
17
19
26
28
88
400

In the case above, the numbers are not in order. With **Tool Renumbering** we can renumber the tools in numeric order. The tools can also be renumbered from a specific number that you choose.

1. Activate this option by choosing the command: **Tool Renumbering**.


Tool List
Permanent Tool
Add Tool
Copy Tool
Delete Tool
Tools Renumbering...
Import Tools...
Update to Machine Tool table...
Export all tools to Excel
Import all tools from Excel

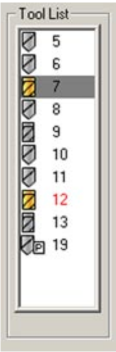
2. You are prompted to enter the new number for the first tool.


Tool Renumbering

First number: 5

OK
Cancel

All the tools are then renumbered accordingly, starting from the number of this first tool.

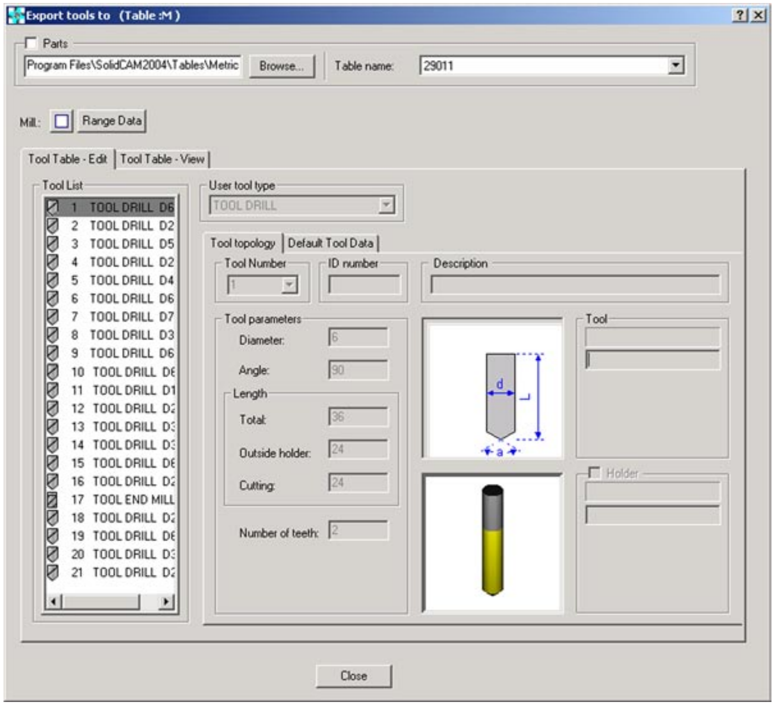

Tool List
5
6
7
8
9
10
11
12
13
19

3. Re-calculate the CAM-Part. The CAM-Part must be recalculated so that the G-Code will have the correct tool number for each job.

A tool that has been marked as a **permanent tool** will not be renumbered.

Import Tool

This option enables you to import tools to the Tool Table from the Current Tool Table or any other Tool table. The Tool tables list will be displayed. The Current Tool Table, which is the first in the list, will be automatically chosen.


Export tools to (Table :M)
☐ Parts
Browse...
Mill:
Range Data
Tool Table - Edit
Tool Table - View
Tool List
User tool type
Tool topology
Default Tool Data
Tool
☐ Holder
d
L
a
Close

Program Files\SolidCAM2004\Tables\Metric

Table name: 29011

1 TOOL DRILL D6
2 TOOL DRILL D2
3 TOOL DRILL D5
4 TOOL DRILL D2
5 TOOL DRILL D4
6 TOOL DRILL D6
7 TOOL DRILL D7
8 TOOL DRILL D3
9 TOOL DRILL D6
10 TOOL DRILL D
11 TOOL DRILL D1
12 TOOL DRILL D
13 TOOL DRILL D
14 TOOL DRILL D
15 TOOL DRILL D
16 TOOL DRILL D
17 TOOL END MILL
18 TOOL DRILL D
19 TOOL DRILL D
20 TOOL DRILL D
21 TOOL DRILL D

User tool type: TOOL DRILL

Tool Number: 1

ID number:

Description:

Tool parameters

Diameter: 6

Angle: 90

Length

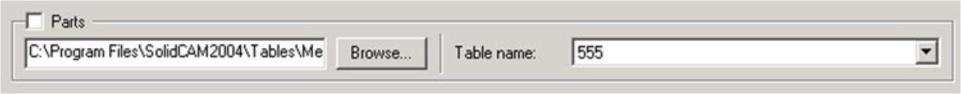
Total: 36

Outside holder: 24

Cutting: 24

Number of teeth: 2

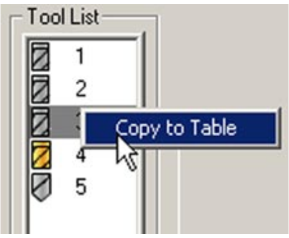
SolidCAM enables you to choose the location of the Tool Tables for the tools import.


☐ Parts
C:\Program Files\SolidCAM2004\Tables\Me
Table name: 555

The **Table Name** combo-box enables you to choose the Tool table for the tools import.

The **Parts** checkbox enables you to import tools from the CAM-Parts situated in the specified location.

If the **Current Tool Table** is defined, it will be automatically chosen as origin for tools imports.



The tools list displays all the tools contained in the specified tool table. The right-click menu is available.

The **Copy to Table** command imports the selected tools to the Tool Table.

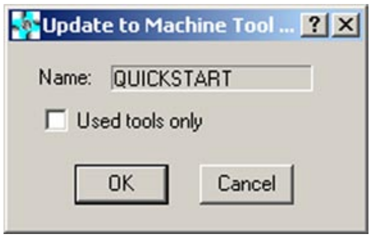
Update to Machine Tool table

This option enables you to update the Machine tool Table with the tools of an active Tool Table.

This option is available when the **Tool Table** dialog is loaded outside the Job.

This option is active only when the **Machine Tool Table** is defined in the MAC file.

The **Update to Machine** tool table dialog will be displayed.

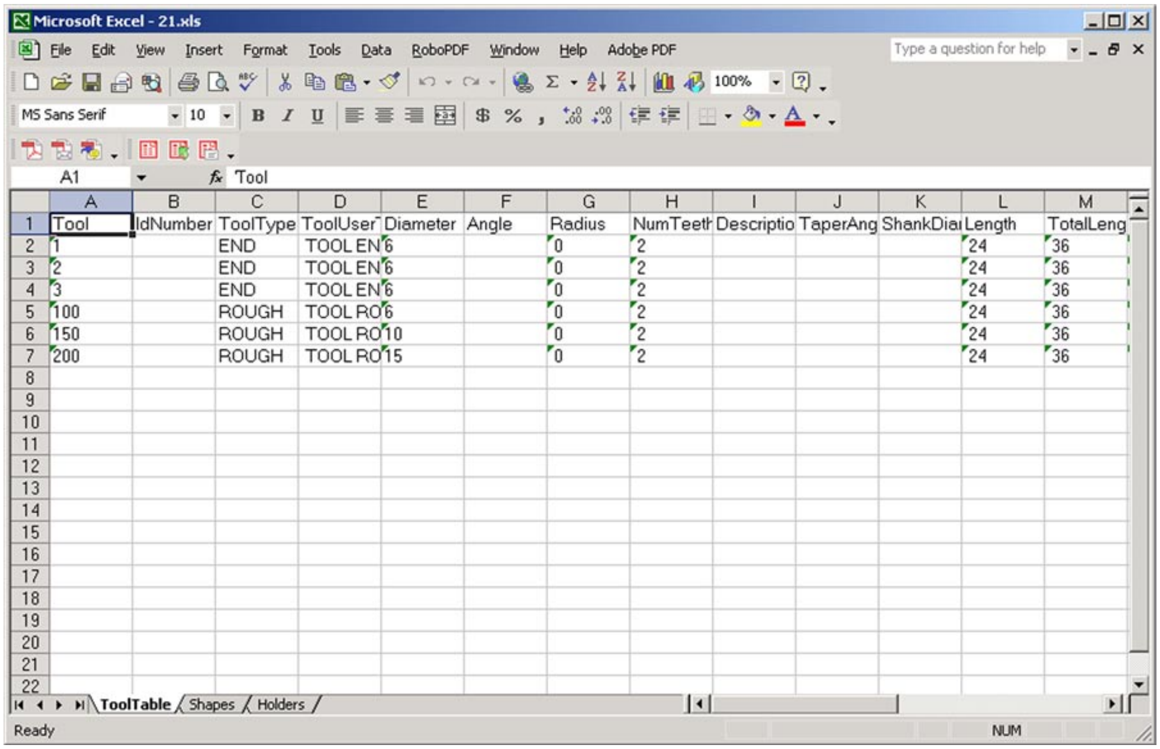


Name: QUICKSTART
☐ Used tools only

This dialog displays the name of the **Machine Tool Table**. The **Used Tools only** option enables you to update the Machine Tool Table only with the tools used in Jobs.

Export all tools to Excel

This option enables you to export all the tool data to an Excel file.


Microsoft Excel - 21.xls
File
Edit
View
Insert
Format
Tools
Data
RoboPDF
Window
Help
Adobe PDF
Type a question for help
100%
MS Sans Serif
10
A1
Tool
ToolTable
Shapes
Holders
Ready
NUM

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Tool	IdNumber	ToolType	ToolUser	Diameter	Angle	Radius	NumTeeth	Descriptio	TaperAng	ShankDia	Length	TotalLeng
2	1		END	TOOL EN	6		0	2				24	36
3	2		END	TOOL EN	6		0	2				24	36
4	3		END	TOOL EN	6		0	2				24	36
5	100		ROUGH	TOOL RO	6		0	2				24	36
6	150		ROUGH	TOOL RO	10		0	2				24	36
7	200		ROUGH	TOOL RO	15		0	2				24	36

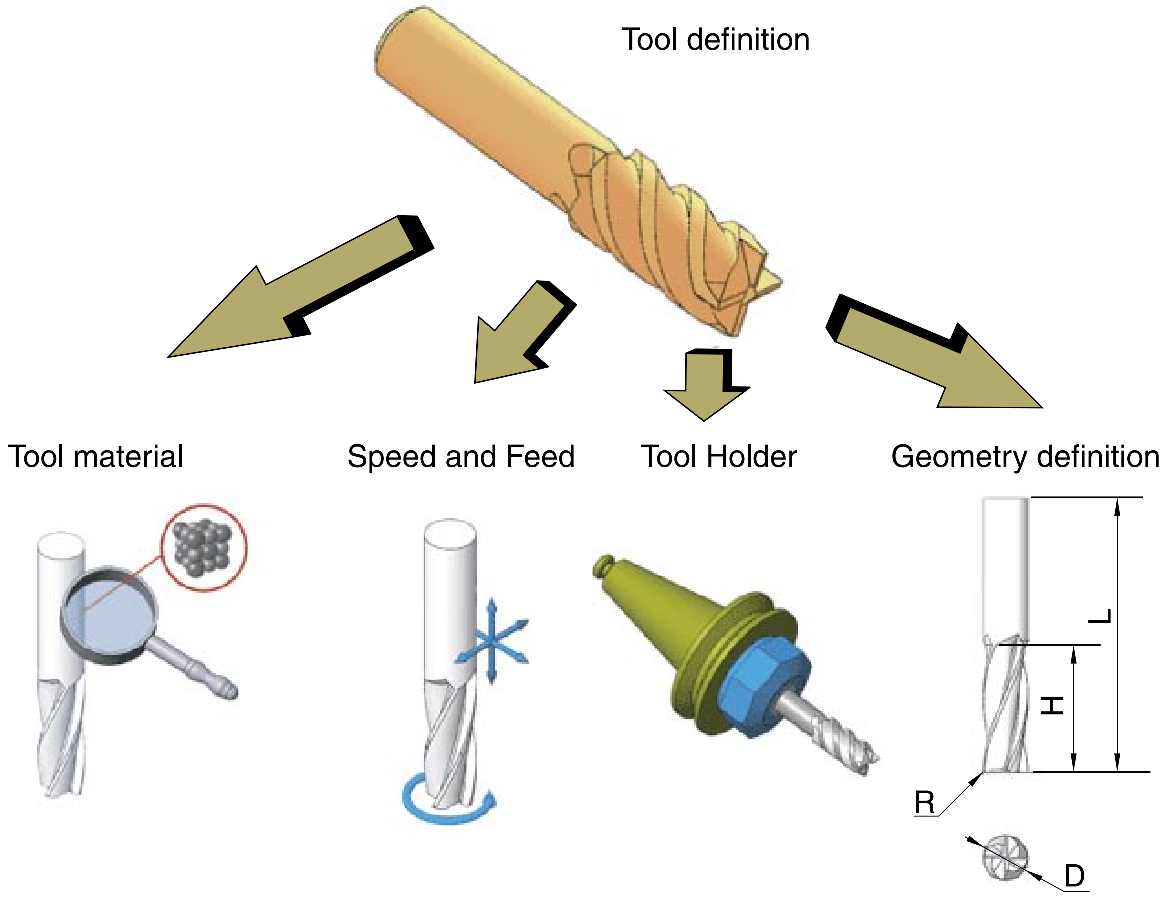
Import all tools from Excel

This option enables you to import the tool data from an Excel file.

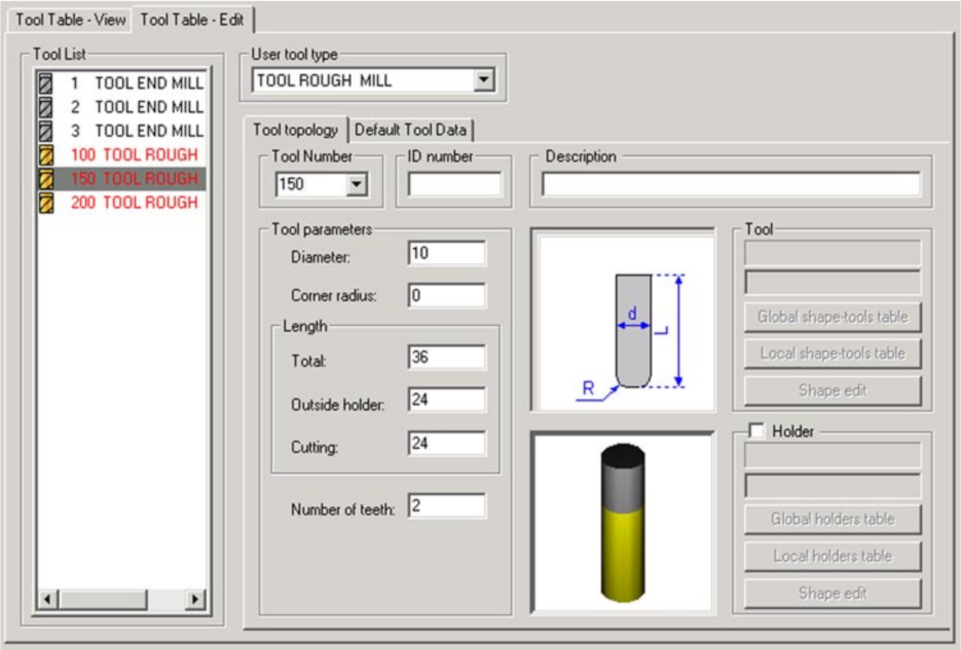
3.8 Tool data

SolidCAM enables you to define the **Tool** with the following data:

- Geometry parameters
- Tool Material
- Feed and Spin defaults
- Tool Holder


Tool definition
Tool material
Speed and Feed
Tool Holder
Geometry definition
L
H
R
D

The **Tool Data** area displays the parameters of the selected tool.


Tool Table - View
Tool Table - Edit
Tool List
Tool topology
Default Tool Data
ID number
Description
Tool
Global shape-tools table
Local shape-tools table
Shape edit
Global holders table
Local holders table
Shape edit
d
L
R

Tool List:
1 TOOL END MILL
2 TOOL END MILL
3 TOOL END MILL
100 TOOL ROUGH
150 TOOL ROUGH
200 TOOL ROUGH

User tool type: TOOL ROUGH MILL

Tool Number: 150

Tool parameters
Diameter: 10
Corner radius: 0
Length
Total: 36
Outside holder: 24
Cutting: 24
Number of teeth: 2

☐ Holder

3.8.1 Tool Type

This field shows the tool type.

The tool types were defined with the Tool type names dialog.

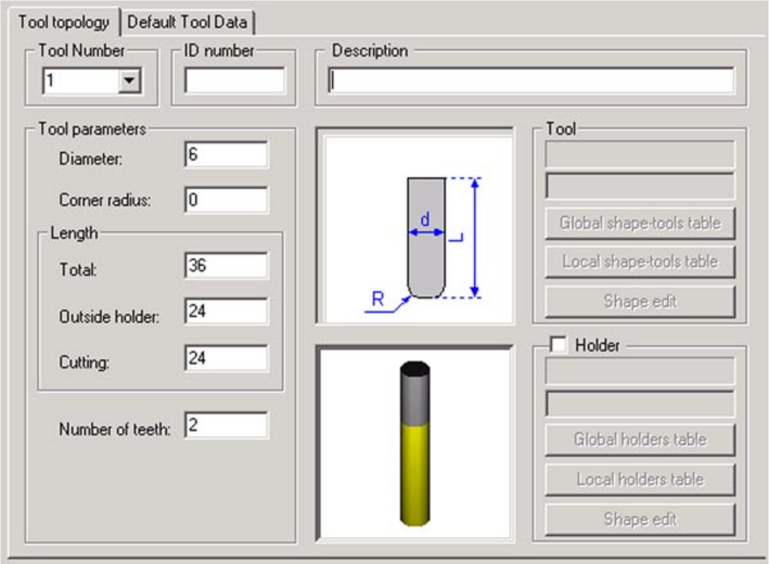
These types are the internal tool types. SolidCAM checks the possibility of a specific internal tool type to work in the selected job.

Selecting a shaped tool will activate the Tool name section on the Tool topology page.

The **Topology page** contains the **Topology Data** of the tool like diameter, length, angle, corner radius etc.

The **Default Tool Data** page contains the default **Tool Material data**, **Feed** and **Spin** data and diameter and length offset numbers.

3.8.2 Topology page


Tool topology
Default Tool Data
Tool
Global shape-tools table
Local shape-tools table
Shape edit
☐ Holder
Global holders table
Local holders table
Shape edit
d
L
R

Tool Number: 1
ID number:
Description:

Tool parameters
Diameter: 6
Corner radius: 0

Length
Total: 36
Outside holder: 24
Cutting: 24

Number of teeth: 2

ID Number

You can use any number, e.g. the catalog number of the tool to facilitate its identification.

Tool Number

Defines the number of the tool in the CAM-Part. SolidCAM will assign a sequential tool number for each new job. You can use previously selected tools by simply entering the tool number. The tool number is used in the G-Code when tool changes, tool length and radius values for tool compensation (G4x) are read from the machine controller. Make sure that the tool number is identical to the actual tool position in the machine before you machine the workpiece.

Diameter

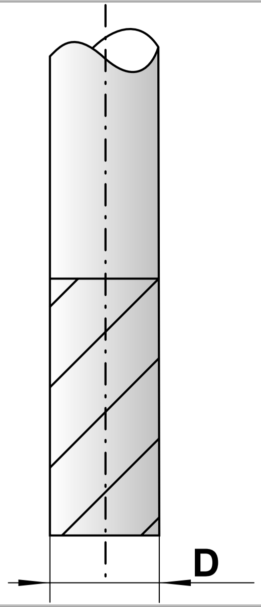
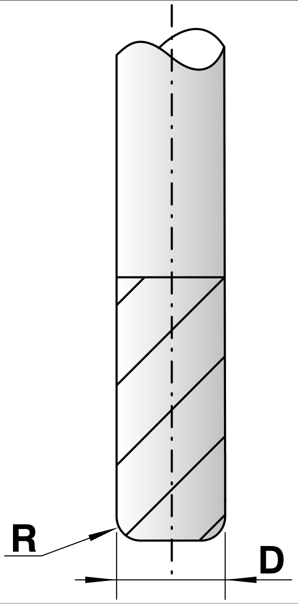
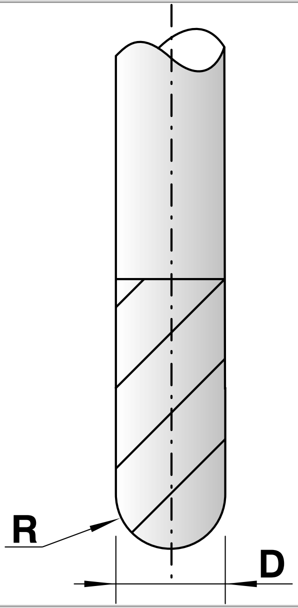
Defines the cutting diameter of the tool.

Corner Radius/Angle

When the tool type is **Rough-Mill** or **End-Mill**, the **Corner Radius** field is displayed. If the tool type is **Drill**, the **Angle** field is displayed.

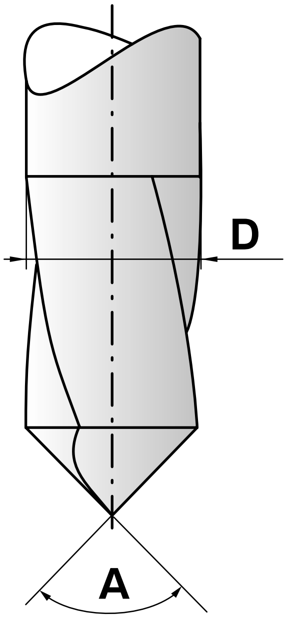
- **Corner Radius**

Enter the corner radius of the tool. There are three possibilities:

		
A **flat mill** has a corner radius of zero.	In **ball nosed mills**, the corner radius is equal to the radius of the tool.	In **radius end mills**, the corner radius is less than the radius of the tool.

- **Angle**

Enter the drill point angle of the tool (between 0.01 and 180 deg).



Length

This field defines the length of the tool. In milling calculations, the system does not use this data; the length of the tool is only output to the G-Code file.

- **Total**

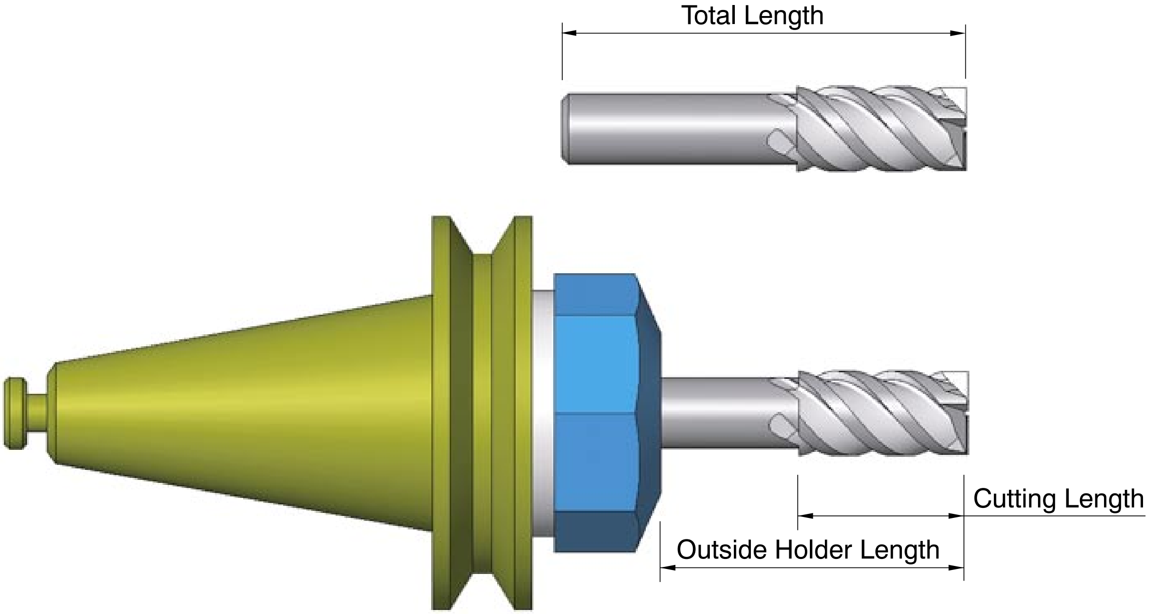
  The total tool length.

- **Outside Holder**

  The length of the tool outside the tool holder.

- **Cutting**

  The length of the cutting part of the tool.



Number of Teeth

Defines the number of teeth of the tool. This field is used when calculating the feed in the Feed rate type FZ.

Description

This field enables you to enter a description of the tool (up to 40 characters).

Tool parameters

This field contains two pictures:

- Schematic picture of the current tool with its dimensions.
- 3D model of the tool with the specified tool holder.

In the graphic area of this page you can rotate, zoom and pan the 3D model of the tool Holder

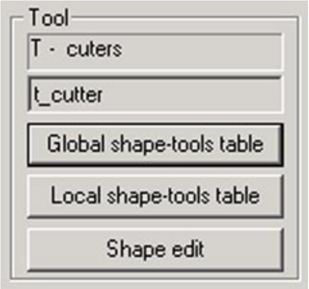
Use the following combinations:

- **Middle mouse button** - Rotation of tool holder
- **Shift + Middle mouse button** - Zoom of tool holder
- **Ctrl + Middle mouse button** - Pan of tool holder

Tool Name

This field enables you to choose the shaped tool from the local/global tool table.

This field is activated only if the shaped tools type (Shaped Tool Drill, Shaped Tool Rough, Shaped Tool End Mill) is chosen.



Tool

T - cuters

t_cutter

- **Global shape-tools table**

The **Shape-tool dialog** will be displayed enabling you to choose the shaped tool from the Global shape-tools table. The selected tool will be copied in the Local shape-tools table that is relative to the current CAM-Part.

- **Local shape-tools table**

This option enables you to choose the previously used in the current CAM-part shape-tool from the Local shape-tools table. The **Shape-tools dialog** will be displayed.

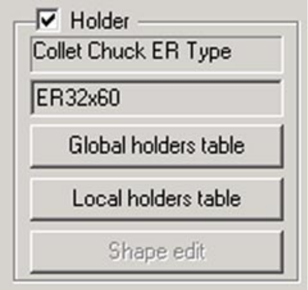
- **Shape Edit**

This option enables you to change dimensions of the selected shape-tool. The **Shape-tools Edit dialog** will be displayed.

For more information on this using shaped tools, please refer to topic **3.10**.

Holder

This field enables you to choose the tool holder parts from the tool holders table.


☑ Holder

Collet Chuck ER Type

ER32x60

The tool adaptor is defined in the MAC-file according to your CNC-machine. If the tool adaptor is not described in the MAC-file, the first tool adaptor from the table will be used.

- **Global holders table**

The **Tool Holders dialog** will be displayed in order to choose the tool holder components from the Global holders table. The selected tool will be copied in the Local holders table that is relative to the current CAM-Part.

- **Local holders table**

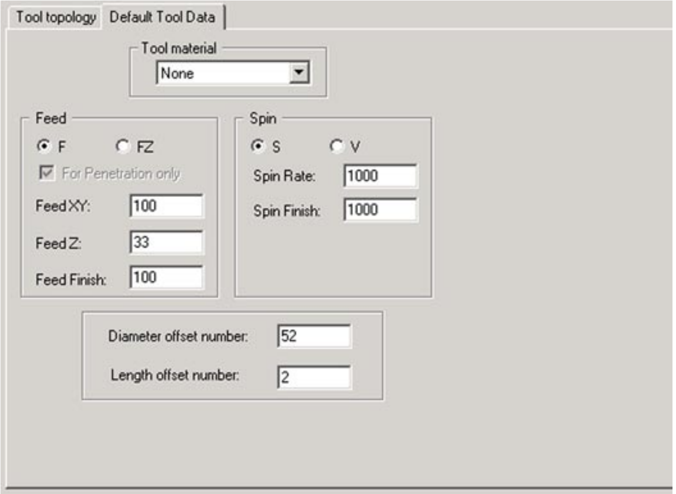
This option enables you to choose the tool holder component previously used in the current CAM-part from the Local holders table. The **Tool Holders dialog** will be displayed

- **Shape Edit**

This option enables you to change dimensions of the selected clamping adaptor. The **Tool Holders Edit dialog** will be displayed.

For more information on this using tool holders, please refer to topic **3.10**.

3.8.3 Default tool data / Job Data



Tool material: None

Feed: ● F ○ FZ

☑ For Penetration only

Feed XY: 100

Feed Z: 33

Feed Finish: 100

Spin: ● S ○ V

Spin Rate: 1000

Spin Finish: 1000

Diameter offset number: 52

Length offset number: 2

Tool Material

The **Tool Material** is used to describe one or more of the following properties of the tool:

- Material of the tool (High Speed Steel, Carbide etc…)
- Working conditions of the tool
- Use of tool (rough, finish, etc…)

The **Tool Material** is used to define the **Speed and Feed defaults**.

When you choose the **Tool material**, the feed type will be changed to **Fz** and the spin type to **V**. The Feed and Spin values are default system values since the **Work material** has not been defined yet. Loading the tool to a job will connect the **Tool material** to the **Work material** and the correct Feed and Spin values will be loaded from the **Speed/Feed defaults** table.

Spin

This field defines the spinning speed of the tool. It defines two spin values:

- **Spin Rate** - Normal spin-rate; used in rough milling.
- **Spin Finish** - Finish spin-rate; used in finish milling.

The spin value can be defined in two types of units, namely **S** and **V**.

S is the default and it signifies **Revolutions per Minute**. **V** signifies Material cutting speed in **Meters/Minute** in the Metric system or in **Feet/Minute** in the Inch system; it is calculated according to the following formula:

V = (S * PI * Tool Diameter) / 1000

Feed

This field defines the feed rate of the tool. It defines three feed values:

- **Feed XY** - Feed-rate in the XY plane.
- **Feed Z** - Feed-rate in the Z direction.
- **Feed Finish** - Feed-rate used for finish milling.

The feed value can be defined in two types of units, namely: **F** and **Fz**.

F is the default and it signifies Units per minute. **Fz** signifies Units per tooth and is calculated according to the following formula:

Fz = F/(Number of Teeth * S)

Diameter Offset Number

This parameter defines the number of the Diameter Offset Register of the current tool in the Offset table of the CNC machine.

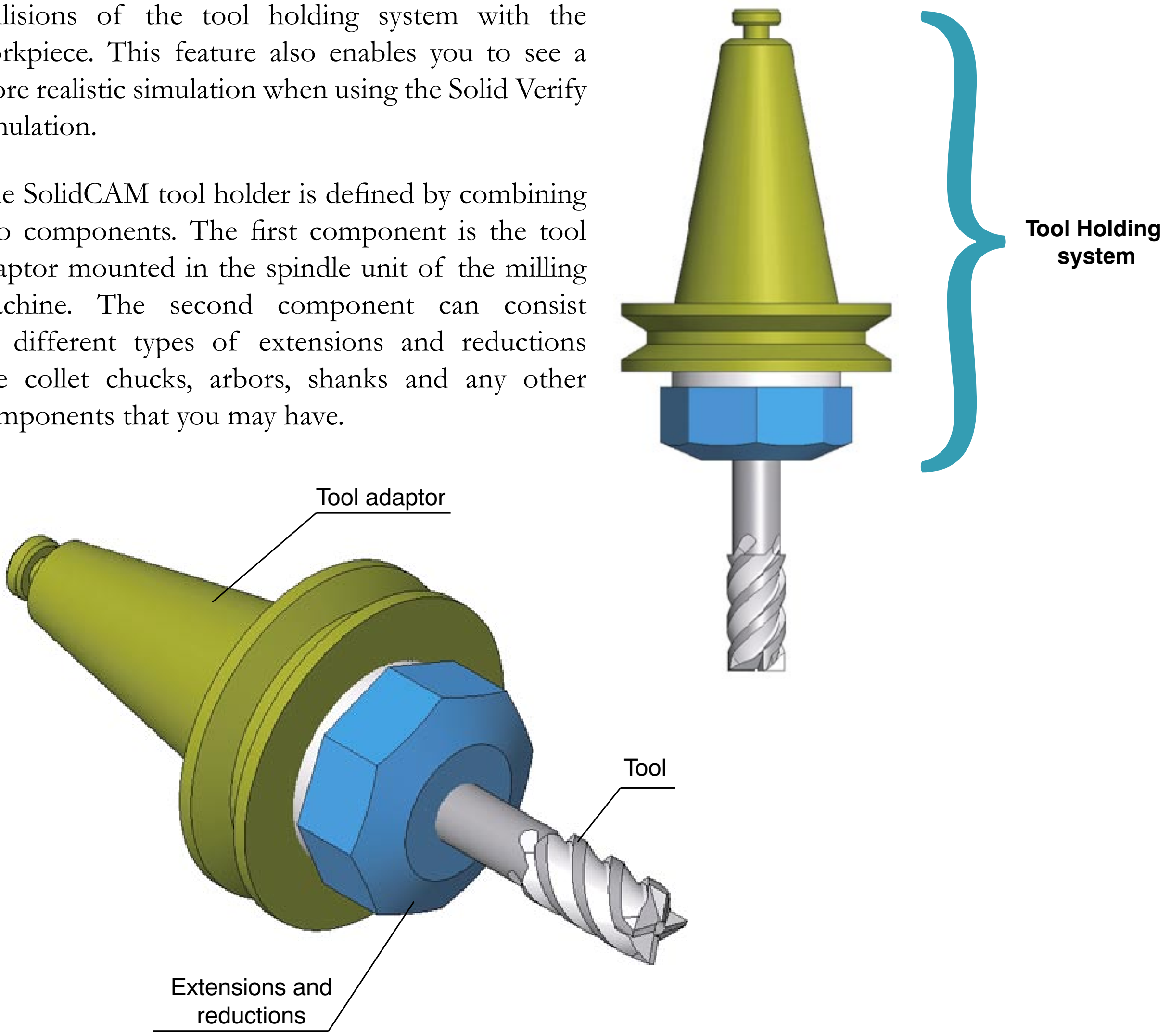
Length Offset number

This parameter defines the number of the Length Offset Register of the current tool in the Offset table of the CNC machine.

3.9 Tool holding system

SolidCAM enables you to define a variety of tool holders to help you check and prevent all possible collisions of the tool holding system with the workpiece. This feature also enables you to see a more realistic simulation when using the Solid Verify simulation.

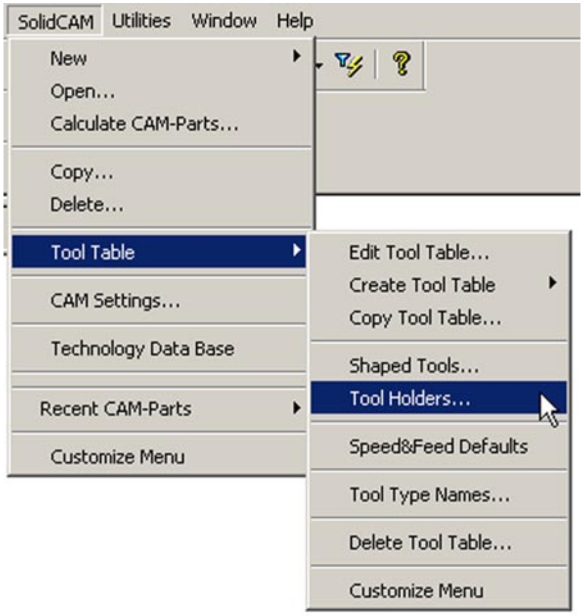
The SolidCAM tool holder is defined by combining two components. The first component is the tool adaptor mounted in the spindle unit of the milling machine. The second component can consist of different types of extensions and reductions like collet chucks, arbors, shanks and any other components that you may have.



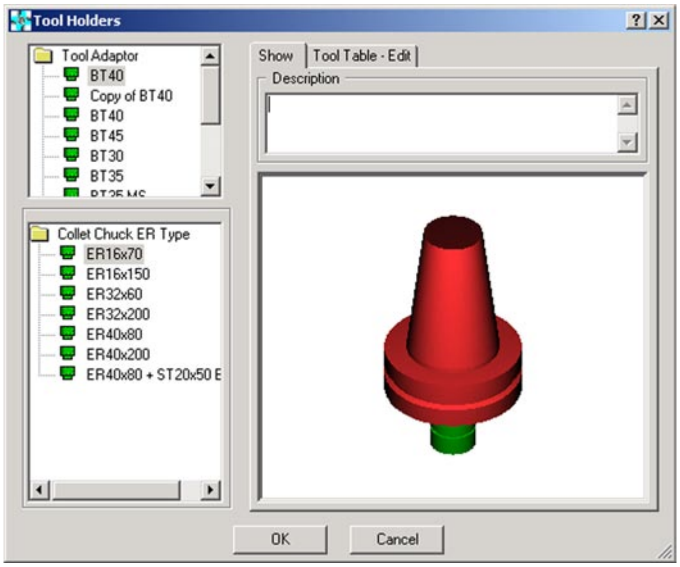
3.9.1 Tool holders dialog

This dialog enables you to define and operate your tool holders.

1. Click on the **SolidCAM** item in the SolidWorks main menu.
2. Choose the **Tool Holders...** item from the **Tool Table** sub-menu.


SolidCAM
Utilities
Window
Help
New
Open...
Calculate CAM-Parts...
Copy...
Delete...
Tool Table
CAM Settings...
Technology Data Base
Recent CAM-Parts
Customize Menu
Edit Tool Table...
Create Tool Table
Copy Tool Table...
Shaped Tools...
Tool Holders...
Speed&Feed Defaults
Tool Type Names...
Delete Tool Table...
Customize Menu

3. The **Tool Holders dialog** will be displayed. A CAD model file will also be displayed to enable you to define the geometry of the new holders.

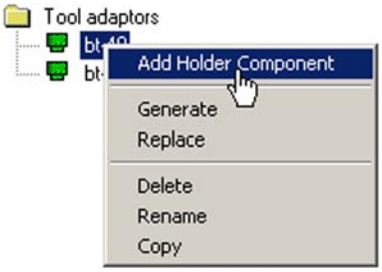

Tool Holders
Tool Adaptor
BT40
Copy of BT40
BT40
BT45
BT30
BT35
Collet Chuck ER Type
ER16x70
ER16x150
ER32x60
ER32x200
ER40x80
ER40x200
ER40x80 + ST20x50 E
Show
Tool Table - Edit
Description
OK
Cancel

Tool holder component tables

The first list displayed above enables you to define and manage your tool adaptors.

1. Right click on the item name in the table.

   The submenu will be displayed.



2. Choose the command from the submenu.

- **Add Holder Component**

  This option enables you to define the geometry of the new tool adaptor that will be added to the table. The **Holder Geometry dialog** will be displayed.

- **Generate**

  This option generates the sketch in the model file with the geometry of the selected tool adaptor.

- **Replace**

  This option enables you to replace the existing tool adaptor geometry with a new one. The **Holder Geometry dialog** will be displayed to select the new geometry.

- **Delete**

  This option enables you to remove the tool adaptor part from the list.

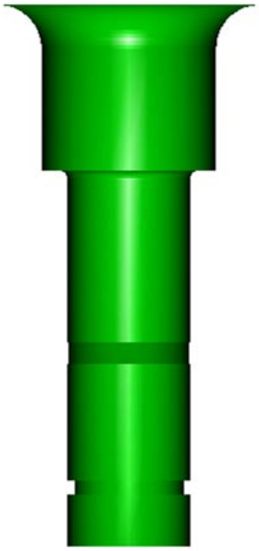
- **Rename**

  This option enables you to assign a new name to the tool adaptor.

- **Copy**

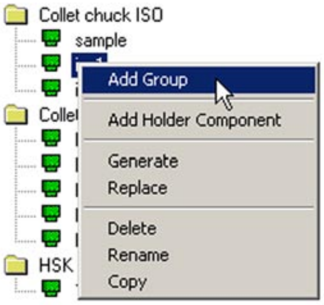
  This option enables you to create a copy of the existing tool adaptor.

The second list enables you to define and manage groups of reductions and extensions such as collet chucks, arbors, shanks etc.



1. Right click on the group/component name in the table.

   The submenu will be displayed.


Collet chuck ISO
sample
Collet
HSK
Add Group
Add Holder Component
Generate
Replace
Delete
Rename
Copy

2. Choose the command from the submenu.

- **Add Group**

  This option enables you to create a group of components and to define separate groups for tool holders components of the same type.

- **Add Holder Component**

  This option enables you to define the geometry of the new component that will be added to the current group of the parts table. The **Holder Geometry dialog** will be displayed.

- **Generate**

  This option generates the sketch in the model file with the geometry of the selected tool holder.

- **Replace**

  This option enables you to replace the existing holder geometry with a new one. The **Holder Geometry dialog** will be displayed to select the new geometry.

- **Delete**

  This option enables you to remove the tool holder component from the list.

- **Rename**

  This option enables you to assign a new name to the tool holder component or group.

- **Copy**

  This option enables you to create a copy of the existing tool holder component.

Description

SolidCAM enables you to type a brief description of the tool holder component (reduction or extension).

Show tab

This page enables you to see a 3D model of the tool holder.

In the graphic area of this page you can rotate, zoom and pan the 3D model of the tool holder.

Use the following combinations:

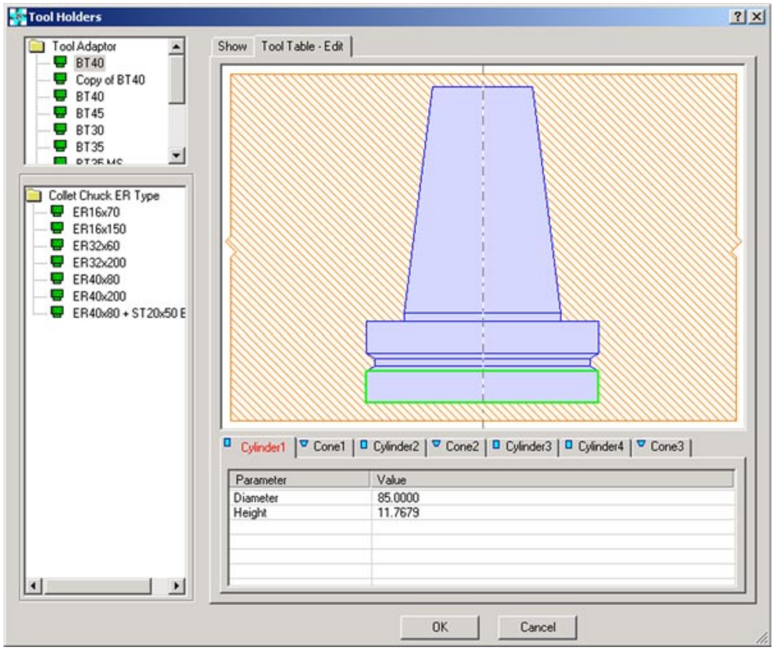
- **Middle mouse button** - Rotation of tool holder
- **Shift + Middle mouse button** - Zoom of tool holder
- **Ctrl + Middle mouse button** - Pan of tool holder

Edit tab

This page enables you to edit the parameters of the selected tool holder component.

1. Select the tool holder component.
2. Switch to the **Edit page** to change the dimensions of the tool holder component.

The interactive picture of the selected tool holder component will be displayed. The parameters pages are located below the picture.


Tool Holders
Tool Adaptor
BT40
Copy of BT40
BT40
BT45
BT30
BT35
Collet Chuck ER Type
ER16x70
ER16x150
ER32x60
ER32x200
ER40x80
ER40x200
ER40x80 + ST20x50 E
Show
Tool Table - Edit
Cylinder1
Cone1
Cylinder2
Cone2
Cylinder3
Cylinder4
Cone3
OK
Cancel

Parameter	Value
Diameter	85.0000
Height	11.7679

SolidCAM divides the tool holder component into solid segments like cylinders, cones or toruses. For each solid primitive, SolidCAM determines actual dimensions.

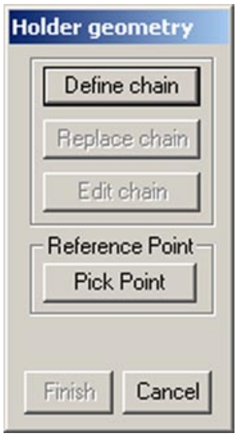
For more information on this subject, please refer to topic **3.9.3**.

You can change the dimensions of each segment with its specific page.

1. Select the segment on the picture or through the tabs below the picture.
2. Type the new value of the parameter in the value field.

3.9.2 Tool Holder Geometry definition

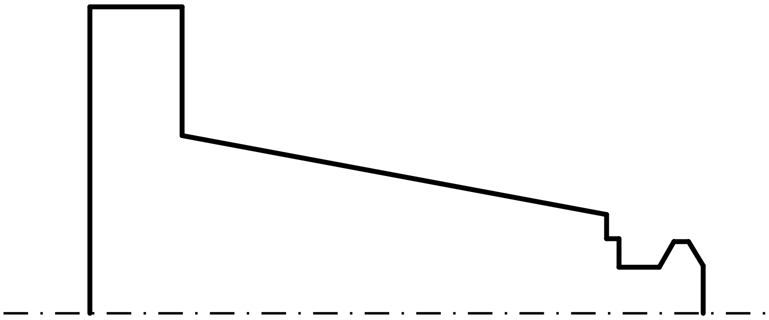
This dialog enables you to define the geometry of the tool holder component.


Holder geometry
Define chain
Replace chain
Edit chain
Reference Point
Pick Point
Finish
Cancel

Define chain

This option enables you to define a new chain of the component geometry. The **Chain selection dialog** will be displayed. In the model file displayed, you can sketch the shape of half the tool holder component and then define the chain. The direction of the chain does not matter.

Note that the left side of the sketch is the bottom of the tool holder component and the right side is the top.



Replace chain

This command enables you to replace a chain in the current chain geometry.

Edit chain

This command enables you to update a chain in the current chain geometry.

Reference point

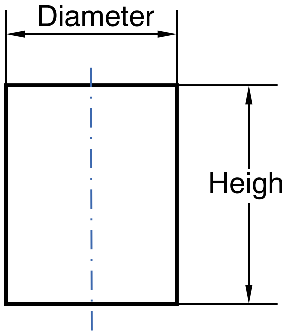
This option enables you to define the reference point through which the rotation axis of the tool holder component passes.

3.9.3 Tool Holder segments

SolidCAM determines three types of segments of the Holder Geometry.

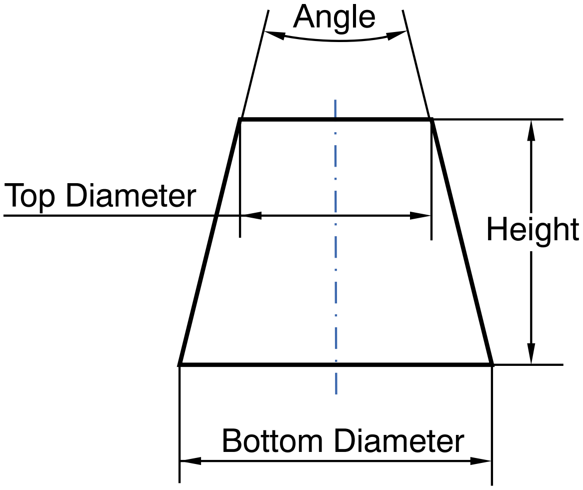
Cylinder

- **Diameter**
- **Height**


Diameter
Height

Cone

- **Top Diameter**
- **Bottom Diameter**
- **Height**
- **Angle**

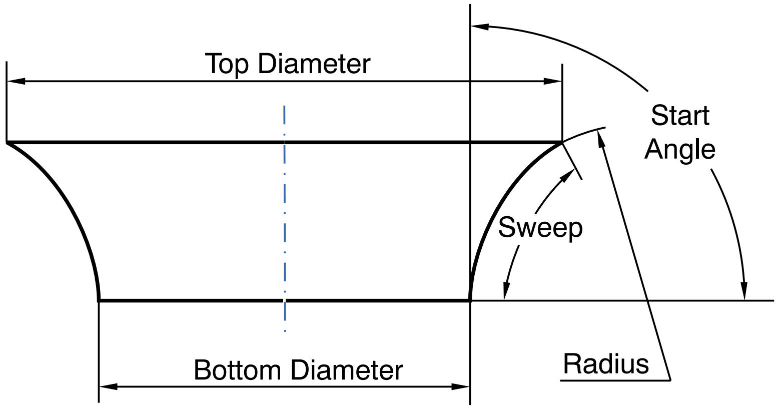

Angle
Top Diameter
Height
Bottom Diameter

The following rules apply when parameters are changed:

- Changing of the **Angle** causes **Top Diameter** change
- Changing of the **Bottom Diameter** causes **Height** change
- Changing of the **Top Diameter** causes **Bottom Diameter** change
- Changing of the **Height** causes **Bottom Diameter** change

Torus

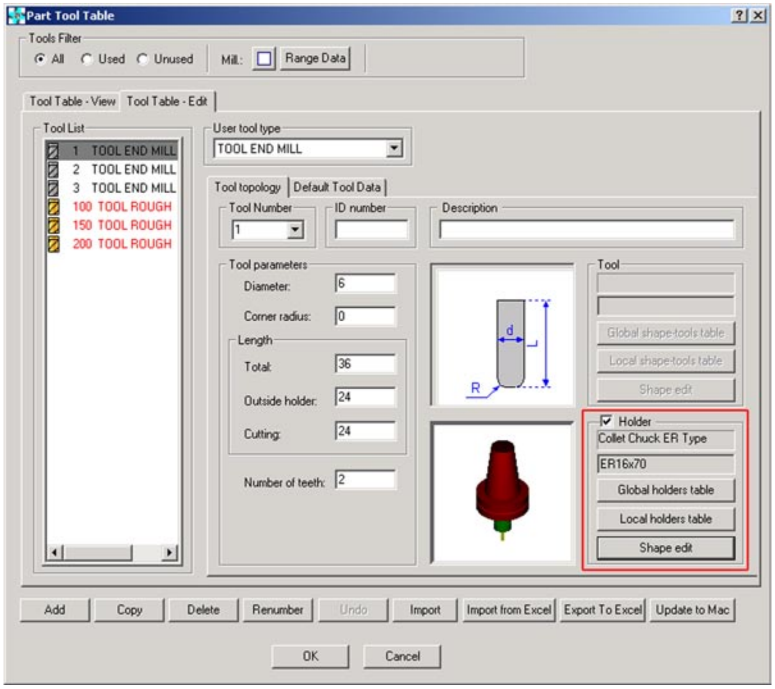
- **Top Diameter**
- **Radius**
- **Start Angle**
- **Sweep**
- **Bottom Diameter**


Top Diameter
Start Angle
Sweep
Bottom Diameter
Radius

The following rules apply when parameters are changed:

- Changing of the **Radius** causes **Bottom Diameter** change
- Changing of the **Top Diameter** causes **Bottom Diameter** change
- Changing of the **Bottom Diameter** causes **Top Diameter** change
- Changing of the **Start Angle** causes **Bottom Diameter** change
- Changing of the **Sweep Angle** causes **Top Diameter** change

3.9.4 Using Tool Holders


Part Tool Table
Tools Filter
All
Used
Unused
Mill:
Range Data
Tool Table - View
Tool Table - Edit
Tool List
1 TOOL END MILL
2 TOOL END MILL
3 TOOL END MILL
100 TOOL ROUGH
150 TOOL ROUGH
200 TOOL ROUGH
User tool type
Tool topology
Default Tool Data
Tool Number
ID number
Description
Tool parameters
Length
d
R
Tool
Global shape-tools table
Local shape-tools table
Shape edit
Holder
Global holders table
Local holders table
Shape edit
Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
Update to Mac
OK
Cancel

Tools Filter: ● All ○ Used ○ Unused

User tool type: TOOL END MILL

Tool Number: 1

Diameter: 6

Corner radius: 0

Total: 36

Outside holder: 24

Cutting: 24

Number of teeth: 2

☑ Holder

Collet Chuck ER Type

ER16x70

- **Global holders table**

The **Tool Holders dialog** will be displayed in order to choose the tool holder components from the Global holders table. The selected tool will be copied in the Local holders table that is relative to the current CAM-Part.

- **Local holders table**

This option enables you to choose the tool holder component previously used in the current CAM-part from the Local holders table. The **Tool Holders dialog** will be displayed

- **Shape Edit**

This option enables you to change dimensions of the selected clamping adaptor. The **Holders Edit dialog** will be displayed.

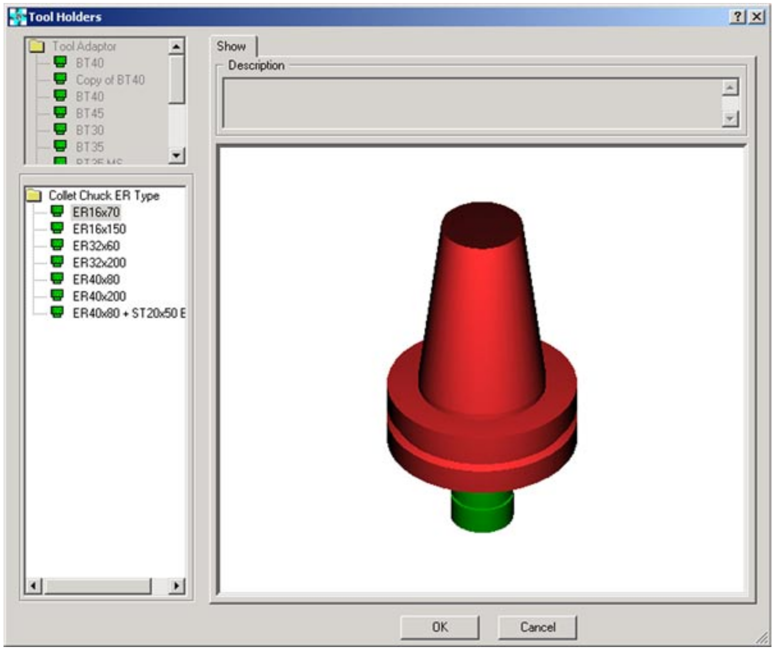
SolidCAM enables you to choose the tool holder component through the **Tool Holders dialog**.

- Note that the tool-adaptor section is inactive and you cannot select the adaptor type. SolidCAM enables you to define the Tool adaptor in the Mac file with the following string:

  mac_holder = BT45

  If the parameter is not defined in the MAC file, SolidCAM will use the first adaptor from the list; the extenders are the same for all the adaptors.

- If all your machines use the same type of adaptor, you do not need to define the mac_holder parameter. In the table, drag and drop the requested adaptor to the first place on the list.


Tool Holders
Tool Adaptor
BT40
Copy of BT40
BT40
BT45
BT30
BT35
Collet Chuck ER Type
ER16x70
ER16x150
ER32x60
ER32x200
ER40x80
ER40x200
ER40x80 + ST20x50 E
Show
Description
OK
Cancel

In the Show area, SolidCAM displays a 3D model of the selected tool holder.

In the graphic area of this page you can rotate, zoom and pan the 3D model of the tool holder.

Use the following combinations:

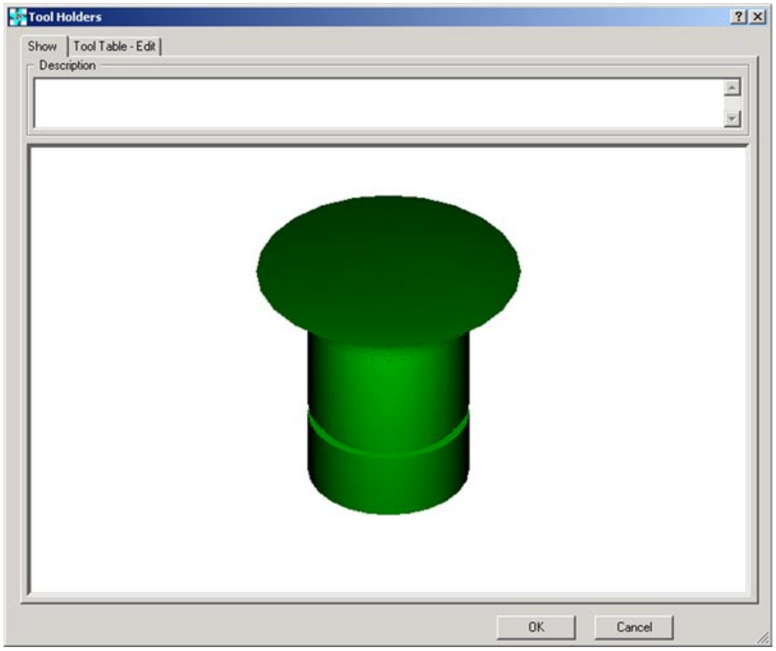
- **Middle mouse button** - Rotation of tool holder
- **Shift + Middle mouse button** - Zoom of tool holder
- **Ctrl + Middle mouse button** - Pan of tool holder

Holders Edit dialog

The **Holders Edit dialog** enables you to edit the dimensions of the selected tool holder component. This dialog contains two tabs:

Show

This page displays a 3D-model of the selected tool holder component.

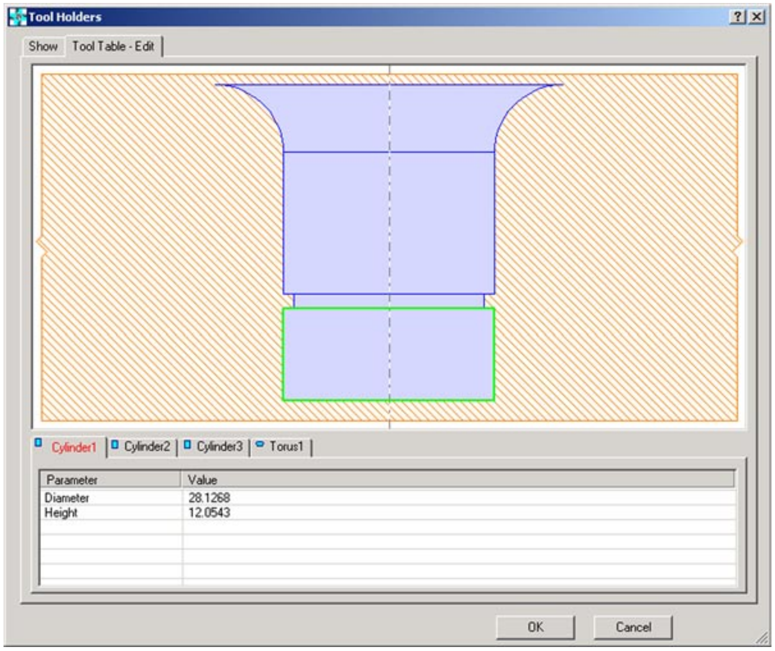


In the graphic area of this page you can rotate, zoom and pan the 3D model of the tool holder.

Use the following combinations:

- **Middle mouse button** - Rotation of tool holder
- **Shift + Middle mouse button** - Zoom of tool holder
- **Ctrl + Middle mouse button** - Pan of tool holder.

Edit


Tool Holders
Show
Tool Table - Edit
Cylinder1
Cylinder2
Cylinder3
Torus1
OK
Cancel

Parameter	Value
Diameter	28.1268
Height	12.0543

The interactive picture of the selected tool holder component will be displayed. The tool holder component is divided into segments such as cylinders, cones and toruses.

The Parameters table below the picture enables you to edit the actual dimensions of the tool holder component.

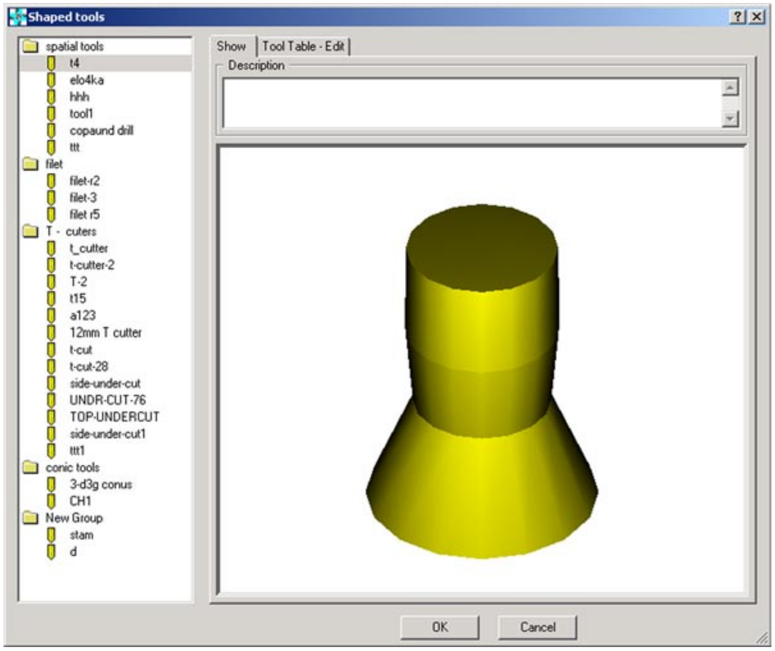
1. Select the holder's element on the picture or through the parameter pages.
2. Type the new value of the parameter in the value field.

3.10 Shaped Tools

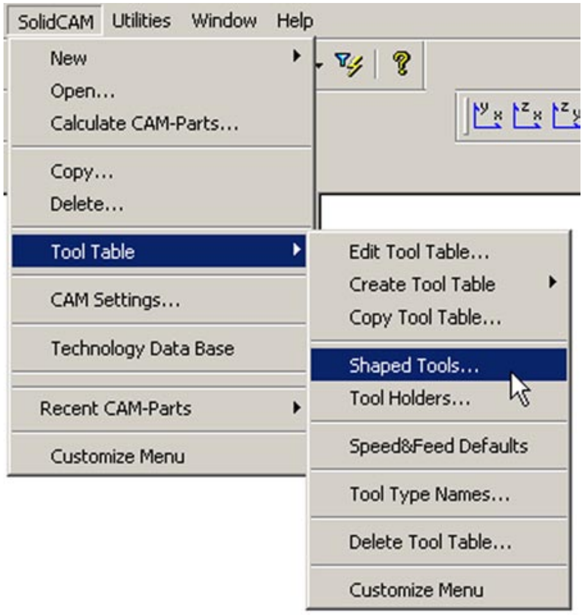
In addition to three built-in **tool types**, SolidCAM enables you to use shaped tools. The geometry of these tools is defined with a sketch. This way you can create your own database of tapered mills, T-shaped mills etc.

3.10.1 Shape-Tools dialog

This dialog enables you to define and operate your shaped tools.


Shaped tools
spatial tools
t4
elo4ka
hhh
tool1
copaund drill
ttt
filet
filet-r2
filet-3
filet r5
T - cuters
t_cutter
t-cutter-2
T-2
t15
a123
12mm T cutter
t-cut
t-cut-28
side-under-cut
UNDR-CUT-76
TOP-UNDERCUT
side-under-cut1
ttt1
conic tools
3-d3g conus
CH1
New Group
stam
d
Show
Tool Table - Edit
Description
OK
Cancel

Choose the **SolidCAM** item from the SolidWorks main menu. Choose the **Shaped tools** item from the sub-menu.


SolidCAM
Utilities
Window
Help
New
Open...
Calculate CAM-Parts...
Copy...
Delete...
Tool Table
CAM Settings...
Technology Data Base
Recent CAM-Parts
Customize Menu
Edit Tool Table...
Create Tool Table
Copy Tool Table...
Shaped Tools...
Tool Holders...
Speed&Feed Defaults
Tool Type Names...
Delete Tool Table...
Customize Menu

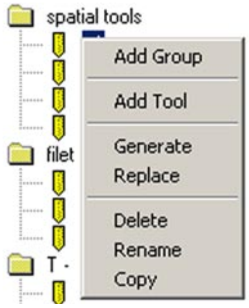
The **Shape-Tools dialog** will be displayed. A CAD model file will also be displayed to enable you to define the geometry of new shape-tools.

Tools list

This field displays the database of shaped tools defined in SolidCAM. With this list you can define and manage shaped tools.

1. Right-click on the any item in the browser field of the **Shape-Tools dialog**.

   The following menu will be displayed.



2. Choose the specific command from the menu.

- **Add Group**

  This option enables you to create a group of the shaped tools. This option enables you to define separate groups for shaped tools of the same type.

- **Add Tool**

  This option enables you to define the geometry of the newly shaped tool that will be added to the current group of the shape-tools table. The **Shape-Tool Geometry dialog** will be displayed.

- **Generate**

  This option generates a sketch in the model file with the geometry of the selected shape-tool.

- **Replace**

  This option enables you to replace the existing shape-tool geometry with a new one. The **Shape-Tool Geometry dialog** will be displayed to select the new geometry.

- **Delete**

  This option enables you to remove the shape-tool from the database.

- **Rename**

  This option enables you to assign a new name to the selected shape-tool or group.

- **Copy**

  This option enables you to create a copy of the selected shape-tool.

Description

You can place a brief description of the shaped tool here.

Show tab

This page enables you to see a 3D model of the selected shaped tool.

In the graphic area of this page you can rotate, zoom and pan the 3D model of the shaped tool.

Use the following combinations:

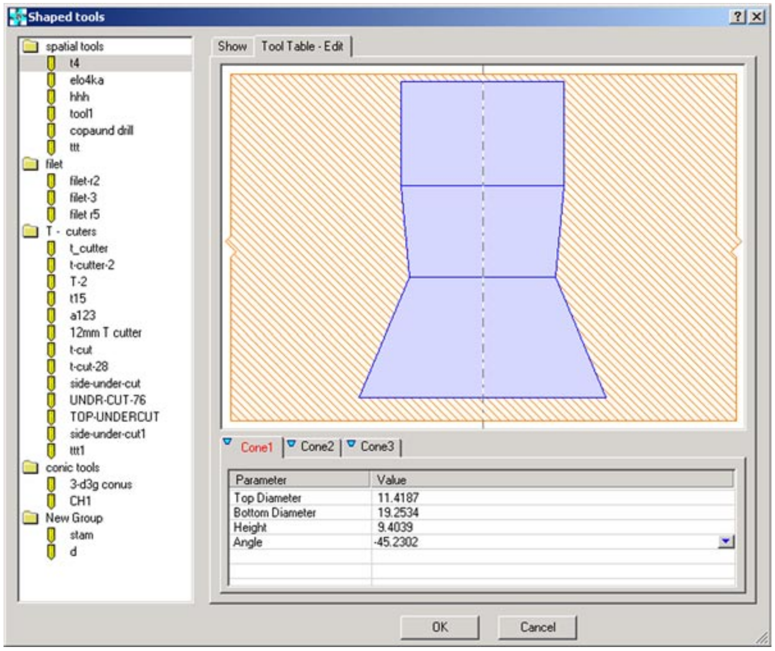
- **Middle mouse button** - Rotation of tool
- **Shift + Middle mouse button** - Zoom of tool
- **Ctrl + Middle mouse button** - Pan of tool

Edit tab

This page enables you to edit the parameters of the selected shaped tool.

1. Select the Shape-tool.
2. Switch to the **Edit page** to change the shape-tool dimensions.

The interactive picture of the selected shaped tool will be displayed. The parameters pages are located below the picture.


Shaped tools
spatial tools
t4
elo4ka
hhh
tool1
copaund drill
ttt
filet
filet-r2
filet-3
filet r5
T - cuters
t_cutter
t-cutter-2
T-2
t15
a123
12mm T cutter
t-cut
t-cut-28
side-under-cut
UNDR-CUT-76
TOP-UNDERCUT
side-under-cut1
ttt1
conic tools
3-d3g conus
CH1
New Group
stam
d
Show
Tool Table - Edit
Cone1
Cone2
Cone3
OK
Cancel

Parameter	Value
Top Diameter	11.4187
Bottom Diameter	19.2534
Height	9.4039
Angle	-45.2302

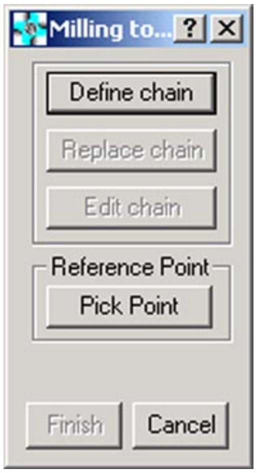
SolidCAM divides the tool into solid segments like cylinders, cones or toruses. For each solid primitive, SolidCAM determines actual dimensions.

You can change the dimensions of each segment with its specific page.

1. Select the tool's element on the picture or through the parameter pages.
2. Type the new value of the parameter in the value field.

3.10.2 Shape-tool geometry definition

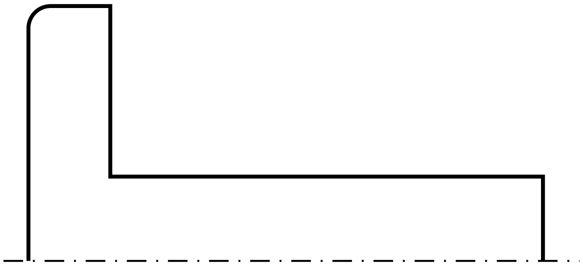
This dialog enables you to define the geometry of the shaped tool.


Milling to...
Define chain
Replace chain
Edit chain
Reference Point
Pick Point
Finish
Cancel

Define chain

This option enables you to define a new chain of the shaped tool geometry. The **Chain selection dialog** will be displayed. In the model file displayed, you can sketch half of the shaped tool and then define the chain. The direction of the chain does not matter.

Note that the left side of the sketch is the bottom of the tool and the right side is the top.



Replace chain

This command enables you to replace a chain in the current chain geometry.

Edit chain

This command enables you to update a chain in the current chain geometry.

Reference point

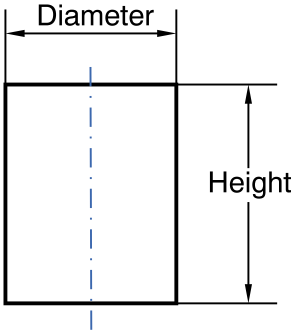
This option enables you to define the reference point through which the rotation axis of the shape-tool passes.

3.10.3 Shaped Tool segments

SolidCAM determines three types of segments of the Shaped Tool Geometry.

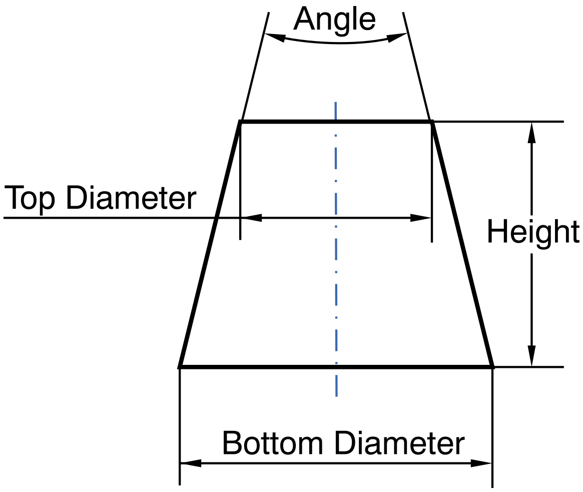
Cylinder

- **Diameter**
- **Height**


Diameter
Height

Cone

- **Top Diameter**
- **Bottom Diameter**
- **Height**
- **Angle**

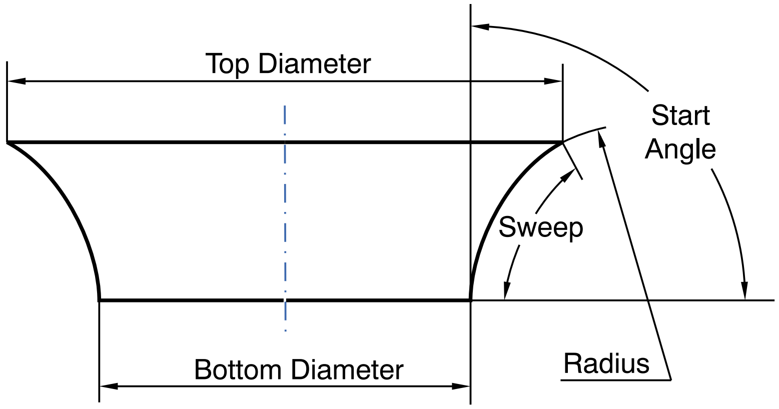

Angle
Top Diameter
Height
Bottom Diameter

The following rules apply when parameters are changed:

- Changing of the **Angle** causes **Top Diameter** change
- Changing of the **Bottom Diameter** causes **Height** change
- Changing of the **Top Diameter** causes **Bottom Diameter** change
- Changing of the **Height** causes **Bottom Diameter** change

Torus

- **Top Diameter**
- **Radius**
- **Start Angle**
- **Sweep**
- **Bottom Diameter**

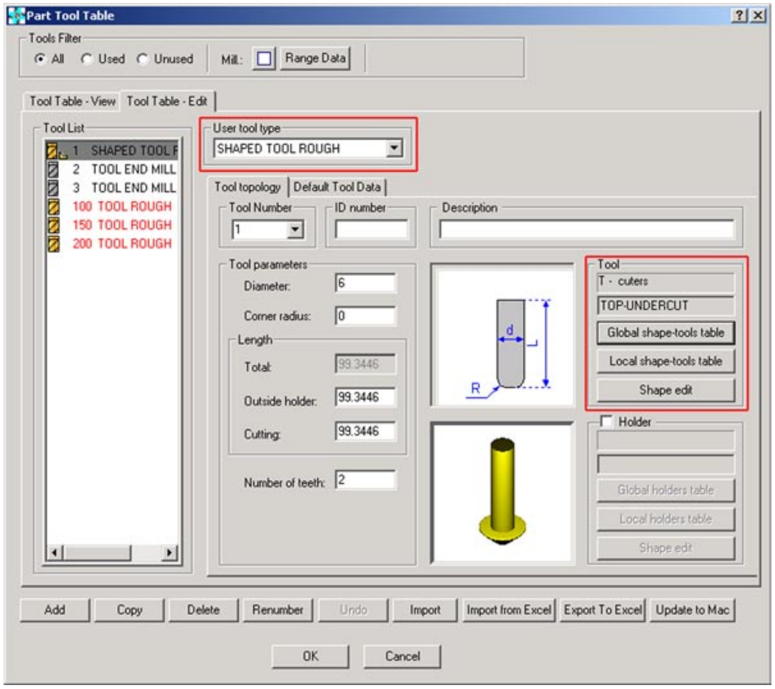

Top Diameter
Start
Angle
Sweep
Radius
Bottom Diameter

The following rules apply when parameters are changed:

- Changing of the **Radius** causes **Bottom Diameter** change
- Changing of the **Top Diameter** causes **Bottom Diameter** change
- Changing of the **Bottom Diameter** causes **Top Diameter** change
- Changing of the **Start Angle** causes **Bottom Diameter** change
- Changing of the **Sweep Angle** causes **Top Diameter** change

3.10.4 Using Shaped tools

The **Tool** field enables you to choose the shaped tool from the local/global tool table.


Part Tool Table
Tools Filter
Mill:
Range Data
Tool Table - View
Tool Table - Edit
Tool List
User tool type
Tool topology
Default Tool Data
Tool Number
ID number
Description
Tool parameters
Length
d
R
Tool
Global shape-tools table
Local shape-tools table
Shape edit
Holder
Global holders table
Local holders table
Shape edit
Add
Copy
Delete
Renumber
Undo
Import
Import from Excel
Export To Excel
Update to Mac
OK
Cancel

Tools Filter: ● All ○ Used ○ Unused

Tool List:
- 1 SHAPED TOOL F
- 2 TOOL END MILL
- 3 TOOL END MILL
- 100 TOOL ROUGH
- 150 TOOL ROUGH
- 200 TOOL ROUGH

User tool type: SHAPED TOOL ROUGH

Tool Number: 1

Diameter: 6

Corner radius: 0

Total: 99.3446

Outside holder: 99.3446

Cutting: 99.3446

Number of teeth: 2

Tool: T - cuters

TOP-UNDERCUT

☐ Holder

This field is activated only if the shaped tools type (Shaped Tool Drill, Shaped Tool Rough, Shaped Tool End Mill) is chosen.

- **Global shape-tools table**

  The **Shape-tool dialog** will be displayed enabling you to choose the shaped tool from the Global shape-tools table. The selected tool will be copied in the Local shape-tools table that is relative to the current CAM-Part.

- **Local shape-tools table**

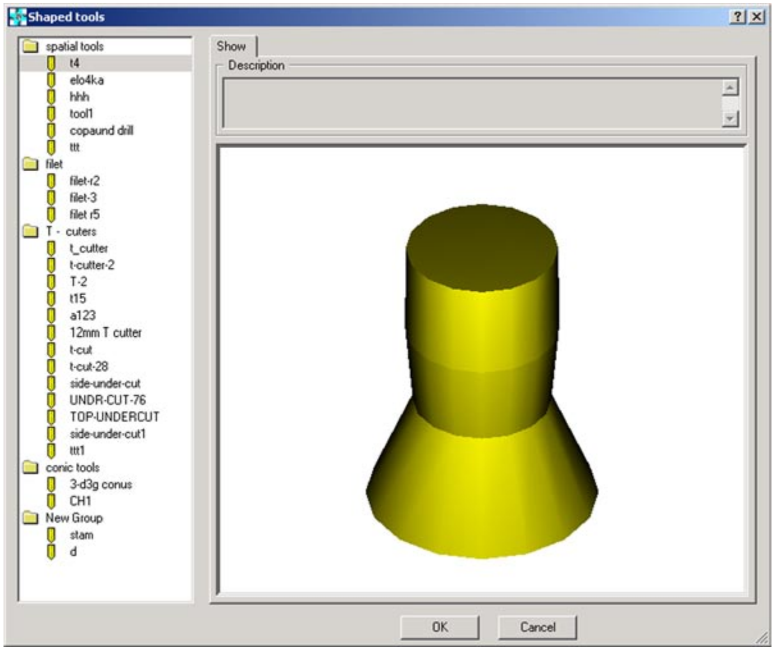
  This option enables you to choose the previously used in the current CAM-part shape-tool from the Local shape-tools table. The **Shape-tools dialog** will be displayed.

- **Shape Edit**

  This option enables you to change dimensions of the selected shape-tool. The **Shape-tools Edit dialog** will be displayed.

Shape-tool dialog

SolidCAM enables you to choose the specific shaped tool with the Shape-tools browser.


Shaped tools
spatial tools
t4
elo4ka
hhh
tool1
copaund drill
ttt
filet
filet-r2
filet-3
filet r5
T - cuters
t_cutter
t-cutter-2
T-2
t15
a123
12mm T cutter
t-cut
t-cut-28
side-under-cut
UNDR-CUT-76
TOP-UNDERCUT
side-under-cut1
ttt1
conic tools
3-d3g conus
CH1
New Group
stam
d
Show
Description
OK
Cancel

In the **Show** area, SolidCAM displays a 3D model of the selected tool.

In the graphic area of this page you can rotate, zoom and pan the 3D model of the tool.

Use the following combinations:

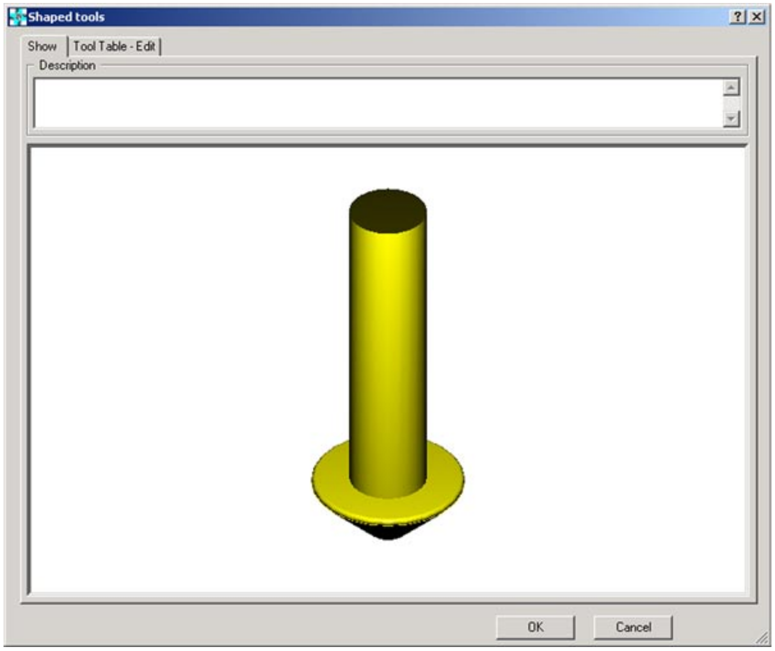
- **Middle mouse button** - Rotation of tool
- **Shift + Middle mouse button** - Zoom of tool
- **Ctrl + Middle mouse button** - Pan of tool

Shape edit

The **Shape-tools Edit dialog** enables you to edit the dimensions of the selected tool. This dialog contains two pages:

Show

This page displays a 3D-model of the selected tool.

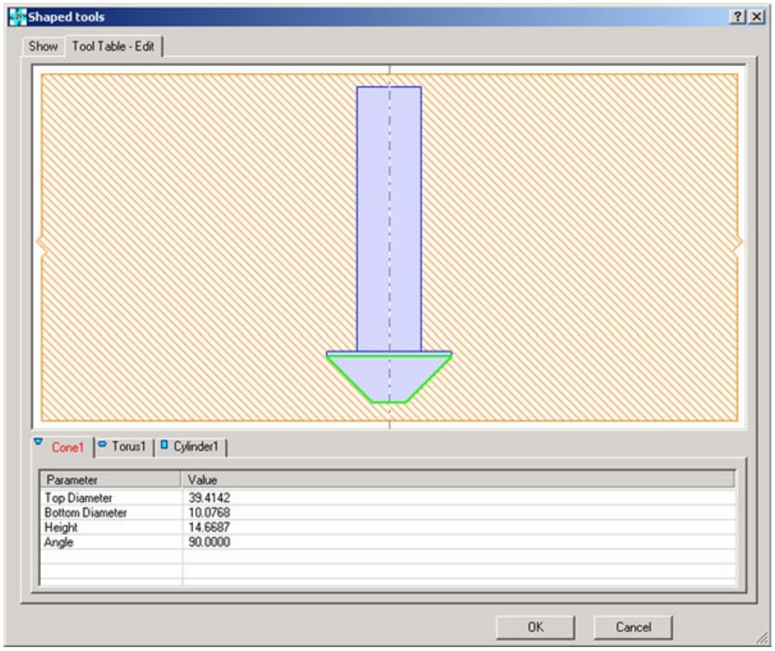

Shaped tools
Show
Tool Table - Edit
Description
OK
Cancel

In the graphic area of this page you can rotate, zoom and pan the 3D model of the tool.

Use the following combinations:

- **Middle mouse button** - Rotation of tool
- **Shift + Middle mouse button** - Zoom of tool
- **Ctrl + Middle mouse button** - Pan of tool

Edit


Shaped tools
?
×
Show
Tool Table - Edit
Cone1
Torus1
Cylinder1
OK
Cancel

Parameter	Value
Top Diameter	39.4142
Bottom Diameter	10.0768
Height	14.6687
Angle	90.0000

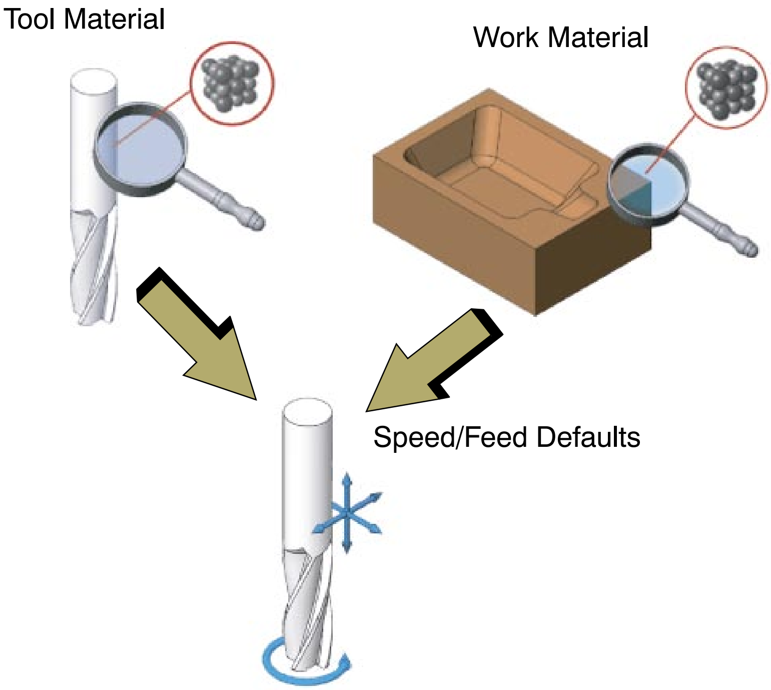
The interactive picture of the selected shaped tool will be displayed. The tool is divided into segments like cylinders, cones and toruses.

With the Parameters table below the picture, you can edit the actual dimensions of the shaped tool.

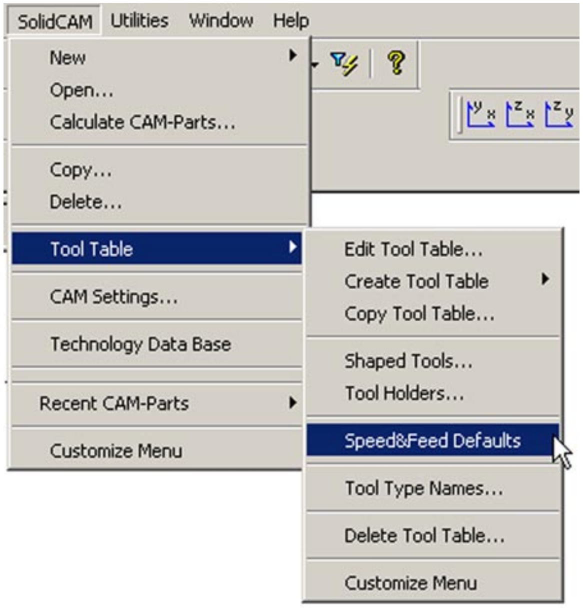
1. Select the tool's element on the picture or through the parameter pages.
2. Type the new value of the parameter in the value field.

3.11 Feed and Speed default

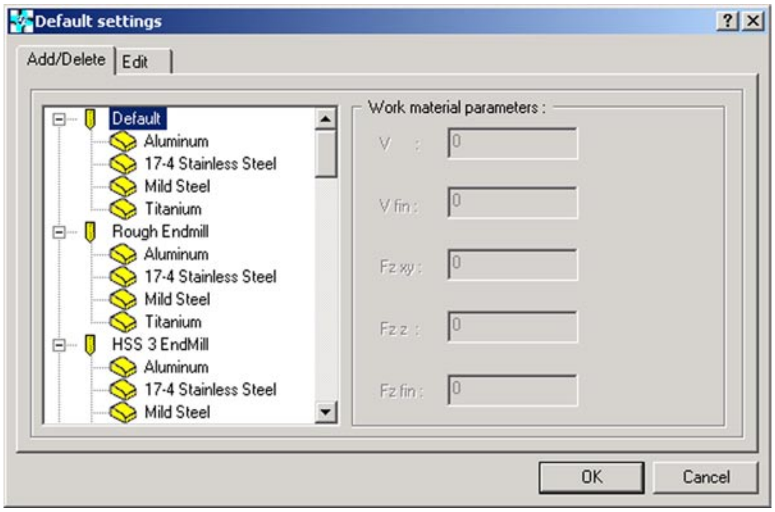
The **Speed/Feed Default** option enables the program to calculate the speed and feed rate according to the **Work material** and **Tool material** you are using.


Tool Material
Work Material
Speed/Feed Defaults

Choose the **Speed&Feed Defaults** item from the **Tool Table** sub-menu.


SolidCAM
Utilities
Window
Help
New
Open...
Calculate CAM-Parts...
Copy...
Delete...
Tool Table
CAM Settings...
Technology Data Base
Recent CAM-Parts
Customize Menu
Edit Tool Table...
Create Tool Table
Copy Tool Table...
Shaped Tools...
Tool Holders...
Speed&Feed Defaults
Tool Type Names...
Delete Tool Table...
Customize Menu

The **Default settings dialog** will be displayed.

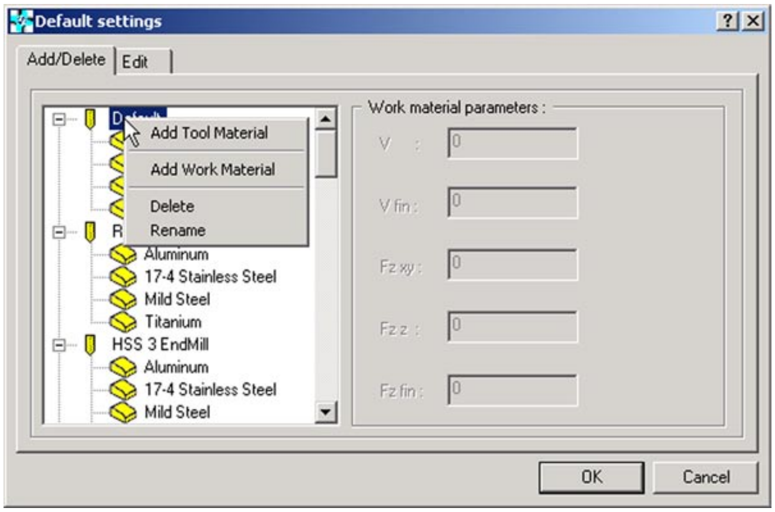

Default settings
Add/Delete
Edit
Default
Aluminum
17-4 Stainless Steel
Mild Steel
Titanium
Rough Endmill
Aluminum
17-4 Stainless Steel
Mild Steel
Titanium
HSS 3 EndMill
Aluminum
17-4 Stainless Steel
Mild Steel
Work material parameters :
V :
V fin :
Fz xy :
Fz z :
Fz fin :
OK
Cancel

V : 0
V fin : 0
Fz xy : 0
Fz z : 0
Fz fin : 0

This dialog enables you to define the **Work Material** and the **Tool Material** and to define the different values of the **Speed and Feed** for each combination set. The dialog shows the tree-control list of the existing Tool and Work Materials. The root items of the tree are the Tool Materials; for each tool material you can see the list of the Work Materials for which Speed/Feed default is defined. The right area of the dialog shows the Speed/Feed parameters for each selected pair of Tool material and Work material.

Add/Delete tab

This tab enables you to operate the Tool and Work materials and define the Speed and feed values for the combination of the tool and work materials.


Default settings
Add/Delete
Edit
Add Tool Material
Add Work Material
Delete
Rename
Aluminum
17-4 Stainless Steel
Mild Steel
Titanium
HSS 3 EndMill
Aluminum
17-4 Stainless Steel
Mild Steel
Work material parameters :
V :
V fin :
Fz xy :
Fz z :
Fz fin :
OK
Cancel

V : 0
V fin : 0
Fz xy : 0
Fz z : 0
Fz fin : 0

Right click in the tool/work material list and choose the specific item from the menu.

- **Add Tool Material**

  This option enables you to add the new Tool Material. The combination pairs of the new **Tool material** with the existing Work materials will be automatically added. Type the name for the new **Tool Material**.

- **Add Work Material**

  This option adds a new Work material. The Work material will be added to all existing Tool material combinations.

- **Delete**

  This option removes the selected Tool/Work material from the list.

- **Rename**

  This option enables you to assign a new name to the selected Tool/Work material.

Work material parameters

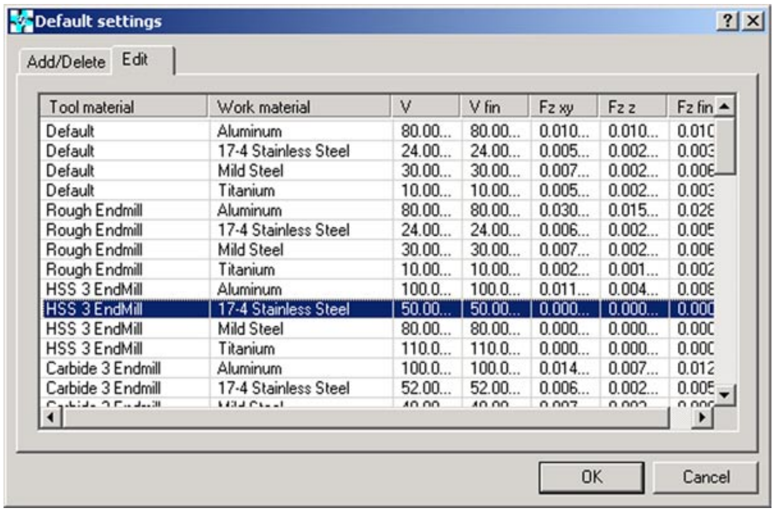
Within this area you have to assign the Speed and Feed values for the selected Tool/Work material combination.

SolidCAM enables you to define the following values:

• **V**	Signifies the Rough Material cutting speed in Meters/Minute in the Metric system or in Feet/Minute in the Inch system.
• **V fin**	Signifies the Finish Material cutting speed in Meters/Minute in the Metric system or in Feet/Minute in the Inch system.
• **Fz XY**	Feed-rate in the **XY** plane, in Units per tooth.
• **Fz Z**	Feed-rate in the **Z** direction, in Units per tooth.
• **Fz fin**	Feed-rate used for finish milling, in Units per tooth.

Edit tab

This tab enables you to edit the values of the Speed and Feed for each combination of the **Tool/Work material**.



Tool material	Work material	V	V fin	Fz xy	Fz z	Fz fin
Default	Aluminum	80.00...	80.00...	0.010...	0.010...	0.01
Default	17-4 Stainless Steel	24.00...	24.00...	0.005...	0.002...	0.00
Default	Mild Steel	30.00...	30.00...	0.007...	0.002...	0.00
Default	Titanium	10.00...	10.00...	0.005...	0.002...	0.00
Rough Endmill	Aluminum	80.00...	80.00...	0.030...	0.015...	0.02
Rough Endmill	17-4 Stainless Steel	24.00...	24.00...	0.006...	0.002...	0.00
Rough Endmill	Mild Steel	30.00...	30.00...	0.007...	0.002...	0.00
Rough Endmill	Titanium	10.00...	10.00...	0.002...	0.001...	0.00
HSS 3 EndMill	Aluminum	100.0...	100.0...	0.011...	0.004...	0.00
HSS 3 EndMill	17-4 Stainless Steel	50.00...	50.00...	0.000...	0.000...	0.00
HSS 3 EndMill	Mild Steel	80.00...	80.00...	0.000...	0.000...	0.00
HSS 3 EndMill	Titanium	110.0...	110.0...	0.000...	0.000...	0.00
Carbide 3 Endmill	Aluminum	100.0...	100.0...	0.014...	0.007...	0.01
Carbide 3 Endmill	17-4 Stainless Steel	52.00...	52.00...	0.006...	0.002...	0.00

Click on the parameter you have to change and enter the new value.

Geometry

4

Whenever you work in CAD, you use 2D wireframe curves and 3D models to describe real life parts. With SolidCAM, you can turn these models into G-Code for any CNC-machine.

A major step in the process from modeling to manufacturing is to tell SolidCAM what and where you want to machine. In SolidCAM you use your existing models to pass this information to the CAM module.

- ❒ Defining a new Geometry
- ❒ Drill Geometry
- ❒ 3D Model Geometry
- ❒ Wireframe Geometries
- ❒ Managing Geometries
- ❒ Geometry synchronization

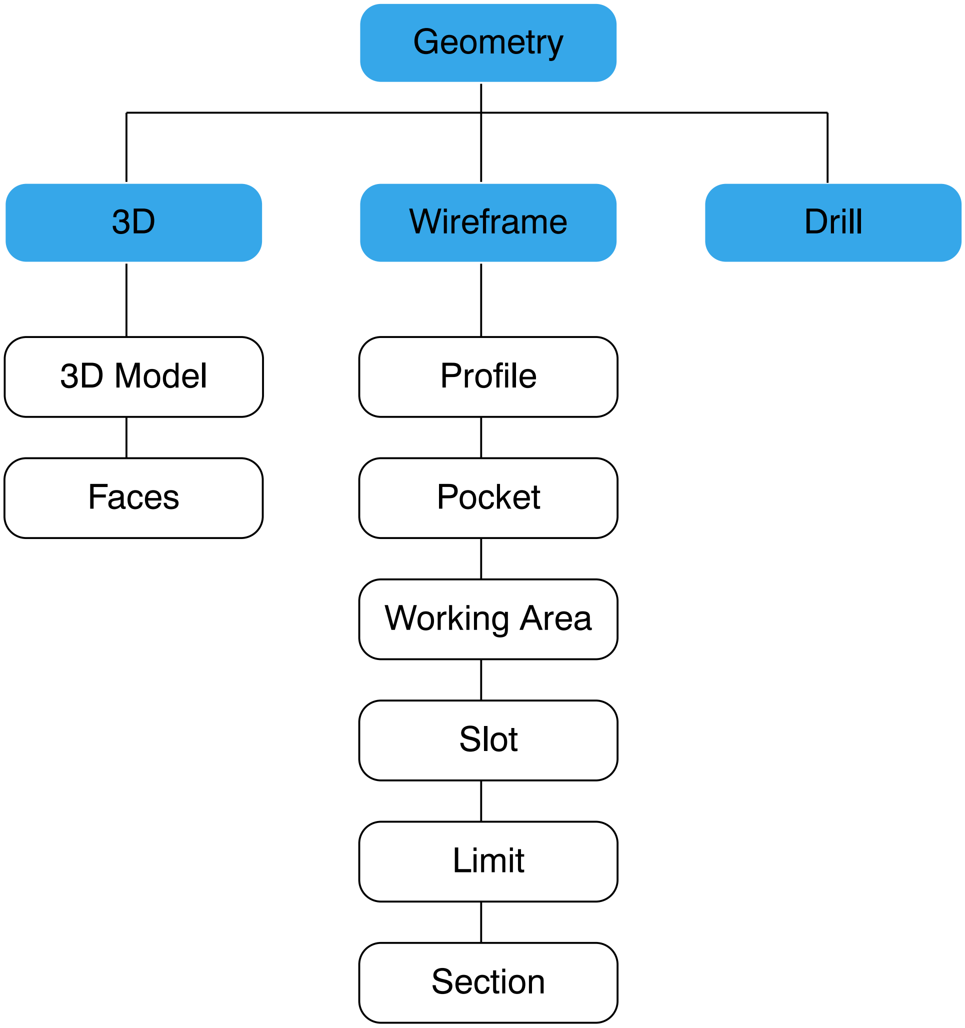
4.1 Introduction

SolidCAM enables you to turn 3D models, 2D and 3D sketches built with SolidWorks into G-code for any CNC-machine. In this process of modelling to manufacturing, geometries have to be defined to determine where the model will be machined.

In SolidCAM, you can define geometries in two places; in the **Geometries** field of the CAM-Manager and in each job. Defining geometries in the Geometries field is only possible if the CAM-part uses homes defined with **Projections mode**; otherwise, this field is inactive. It is recommended to define the geometry from the **Job dialog**; this enables SolidCAM to check if the geometry follows the necessary rules needed in this type of job.

The Geometries defined in SolidCAM are associative to the SolidWorks model. Any change made to the model or sketch will propogate to referenced SolidCAM Geometry.

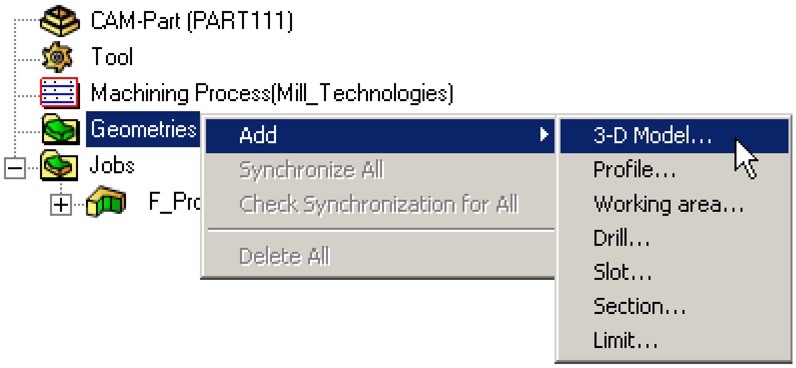
There are types of geometries used in SolidCAM:



4.2 Adding a new Geometry to your CAM-Part

You can add a new geometry from the Geometry field of the CAM-Manager. Whenever there is a geometry required in a Job, you can select it directly with the Define button.

1. In the CAM Manager, move the mouse on the Geometry field and press the **right** mouse button. The **Geometry menu** will be displayed.

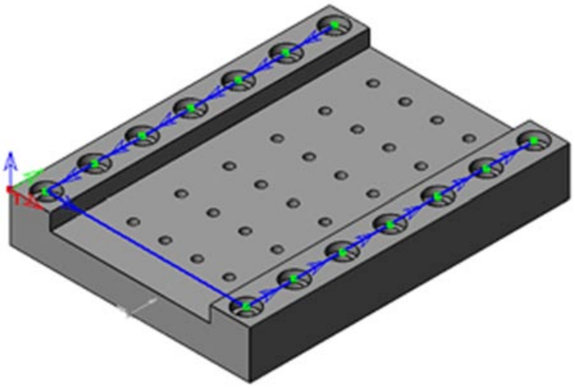

CAM-Part (PART111)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
F_Pro
Add
Synchronize All
Check Synchronization for All
Delete All
3-D Model...
Profile...
Working area...
Drill...
Slot...
Section...
Limit...

2. Move to the **Add** sub-menu and select the **type of geometry** you want to define.

In the Multi-sided mode, the Geometry can be defined only from a Job screen. The reason of this limitation is the dependency of the geometry on the Home that is used for this operation.

4.3 Drill Geometry

A Drill Geometry consists of one or more points (drilling centers) that are defined by X,Y and Z values. They can be selected from models using a number of different methods.

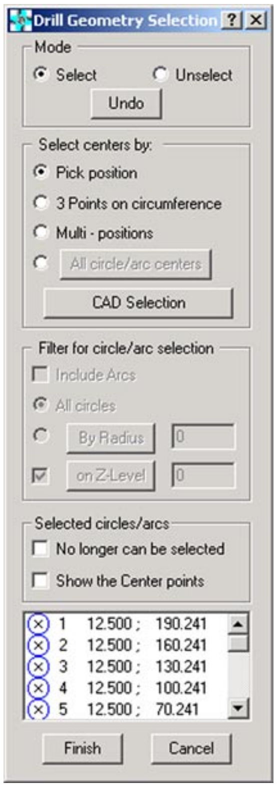


4.3.1 Defining a Drill Geometry

In Drilling operations, you have to define the coordinate points where SolidCAM should execute the drill cycles. The powerful selector tool enables you to define and edit drill positions quickly and easily.

In the **Example** above the drill points have been selected by their radius and Z-level on a 3D solid model. With automatic selection the distance between two drilling points is optimized to reduce machining time.

1. After you have selected the command to add or define a Drill geometry, the **Drill Geometry Selection dialog** is displayed.
2. Select your parameters and options in the **Drill Geometry Selection dialog**.
3. When you have picked all your drill positions, confirm the **Drill Geometry dialog** with **Finish**.
4. Confirm the **Default Geometry Name** (see the topic **10.12**.) or enter a new name for the geometry into the **Geometry name dialog** and confirm with **OK**.


Drill Geometry Selection
Mode
Undo
Select centers by:
3 Points on circumference
Multi - positions
All circle/arc centers
CAD Selection
Filter for circle/arc selection
Include Arcs
By Radius
on Z-Level
Selected circles/arcs
No longer can be selected
Show the Center points
Finish
Cancel

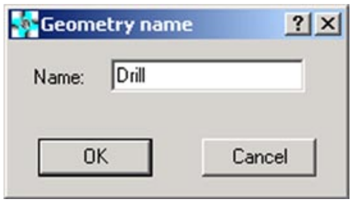
● Select ○ Unselect

● Pick position

● All circles

☑ on Z-Level 0

1	12.500;	190.241
2	12.500;	160.241
3	12.500;	130.241
4	12.500;	100.241
5	12.500;	70.241

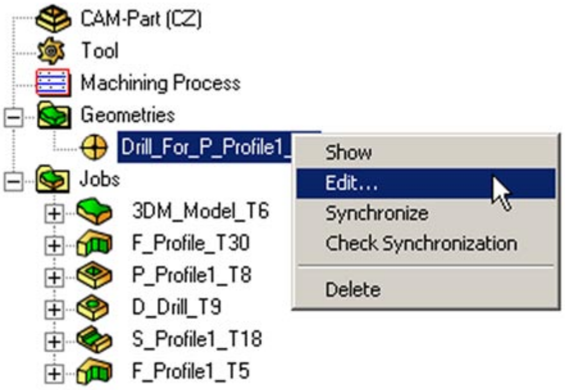

Geometry name
OK
Cancel

Name: Drill

4.3.2 Edit a Drill Geometry

You can add or remove drill positions from previously saved drill geometries.

1. In the **CAM Manager**, right click on a Drill geometry and select Edit from the menu displayed.


CAM-Part (CZ)
Tool
Machining Process
Geometries
Drill_For_P_Profile1_
Jobs
3DM_Model_T6
F_Profile_T30
P_Profile1_T8
D_Drill_T9
S_Profile1_T18
F_Profile1_T5
Show
Edit...
Synchronize
Check Synchronization
Delete

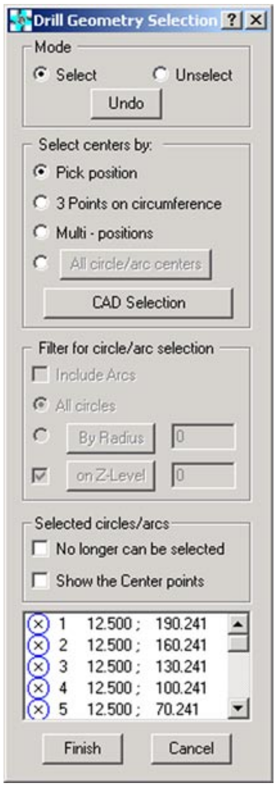
2. The **Drill Geometry Selection dialog** is displayed.

4.3.3 Drill Geometry Selection Dialog

For Drilling operations you have to define the coordinate points where SolidCAM executes the drill cycles.

The powerful selector tool enables you to define and edit drill positions quickly and easily. Whenever you have to define a drill geometry, the **Drill Geometry Selection dialog** will be displayed.

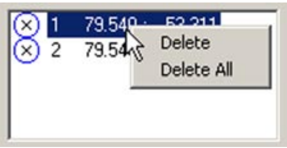
You can switch between the different chain options while selecting drill points. The default option is **Pick position**, but you can start the chain definition with any other option.


Drill Geometry Selection
Mode
● Select ○ Unselect
Undo
Select centers by:
● Pick position
○ 3 Points on circumference
○ Multi - positions
○ All circle/arc centers
CAD Selection
Filter for circle/arc selection
☐ Include Arcs
● All circles
○ By Radius 0
☑ on Z-Level 0
Selected circles/arcs
☐ No longer can be selected
☐ Show the Center points
Finish
Cancel

1	12.500 ;	190.241
2	12.500 ;	160.241
3	12.500 ;	130.241
4	12.500 ;	100.241
5	12.500 ;	70.241

Select/Unselect drill centers

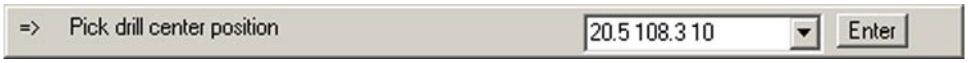
You can switch between the **Select/Unselect** mode to define or remove drill positions from the geometry. The **Undo/Redo** button will delete or restore the last selection. All the selected points will be shown in the list at the bottom of the dialog. To remove a point from the list, right click on the point name in the list and choose the **Delete** option from the menu or choose the **Delete All** option to remove all the points.



Selecting modes (Select drill centers by:)

You can add drill positions to the current geometry. The Selection field offers three different options.

- **Pick position**. You can define Drill positions one by one using the CAD point selection options. You can also type the coordinates (X,Y,Z) into the Edit bar and confirm each drill point by clicking on the Enter button of the Edit bar.

  20.5 108.3 10

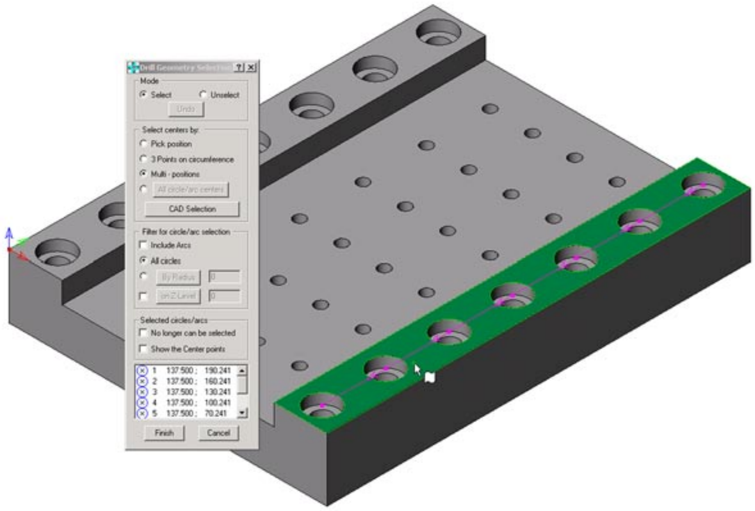
  1. **Pick drill center position** - Select the drill point.
  2. Confirm the selection by clicking on **Enter**.

- **3 Points on circumference**. Usually, all curves and arcs of imported models are converted into splines by the exporting CAD system. Due to the nature of spline curves or surface boundaries, you cannot pick a center position like you could on a circle or arc. SolidCAM will calculate the center position of an arc defined by 3 points positioned on the spline edges. This facilitates selecting drill centers on spline surfaces.

  1. **Pick the first point** - Pick a point on the circumference of the circle or arc using the CAD point selection options or type the X,Y,Z coordinates of the point into the Edit bar and confirm.
  2. **Pick the second point**.
  3. **Pick the third point**.
  4. SolidCAM calculates the center of the three points and displays the drill position.

- **Multi-positions**. You can use the mouse to **drag a box** over the area of the model. SolidCAM will automatically recognize all arcs inside this area and select the center points as drill positions. You can also select the model face; SolidCAM will select all the drill centers on it.

The options you have selected in **Filter** will affect the search when you apply the Multi-positions command.


Drill Geometry Selection
? ×
Mode
● Select
○ Unselect
Undo
Select centers by:
○ Pick position
○ 3 Points on circumference
● Multi - positions
○ All circle/arc centers
CAD Selection
Filter for circle/arc selection
☐ Include Arcs
● All circles
○ By Radius 0
☐ on Z-Level 0
Selected circles/arcs
☐ No longer can be selected
☐ Show the Center points
Finish
Cancel

1 137.500 ; 190.241
2 137.500 ; 160.241
3 137.500 ; 130.241
4 137.500 ; 100.241
5 137.500 ; 70.241

1. Check whether the Filter options you have selected are correct.
2. Click on the screen to select the first diagonal point of the box.
3. Drag a window over the area you want to select drill positions in and click on the screen to pick the second diagonal point.
4. SolidCAM will search the selected box area for drill positions and display the search result.

- **All circle/arc centers**. SolidCAM will search the visible layers for arcs and circles and add all center points as drill positions to the geometry.

The options you have selected in **Filter** will affect the search result when using the **All** command.

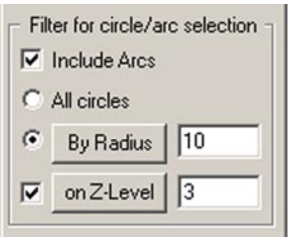
1. Check whether the Filter options you have selected are correct.
2. Click on the **All** button to start the search.
3. SolidCAM will search all visible entities for drill positions and display the search result.

- **CAD selection**. SolidCAM enables you select the drill geometry with the host CAD tools.

1. Check whether the **Filter** options you have selected are correct.
2. Click on the **CAD selection** button. The **Resume** button will be available instead of the **CAD selection** button.
3. Select the geometry with the host CAD tools.
4. Click on the **Resume** button.

Selection filter (Filter for circle/arc selection field)

You can use various filters to search circles/arcs on the model for drill positions. SolidCAM will only select drill positions with the attributes set in the **Filter options**. You can select the following attributes:



Filter for circle/arc selection

☑ Include Arcs

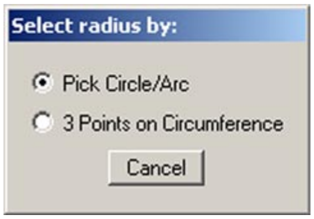
○ All circles

● By Radius 10

☑ on Z-Level 3

- **Include Arcs**. If this option is checked, SolidCAM will also include arc centers in the search for drill positions. This option can solve the occasional problem caused by imported 2D sketches that have circle entities cut to halves. It also enables you to place drills in filleted corners.
- **All Circles**. SolidCAM will search the model for circles. Arcs or broken circles will not be included in the search result.
- **By Radius**. You can limit the search by defining a radius value. Only arcs and circles with the given radius will be selected and their center position will be added to the drill geometry.

You can select a radius value from the model. Click on the **By Radius** button. In the **Select radius by**: dialog you can select how you want to pick the radius from your model. Select the corresponding option and pick the radius from an existing circle/arc or use the 3 Points on circumference definition.



Select radius by:

● Pick Circle/Arc

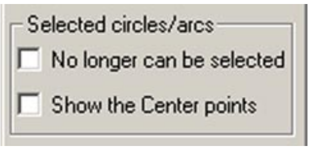
○ 3 Points on Circumference

Cancel

- **Z-Level**. You can also limit the search to a Z-level. Only arcs and circles on the given XY-plane will be selected and their center position will be added to the drill geometry.

You can select a Z-level from your model. Click on the Z-level button and select a point on your model using the Point selection options. The Z-coordinate of the selected point will be passed to the corresponding field.

Selected circles/arcs



Selected circles/arcs

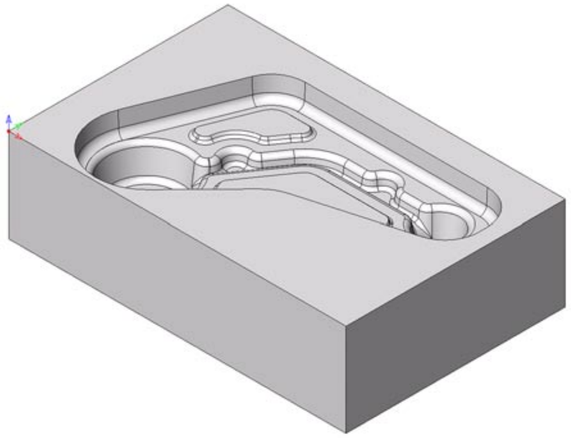
☐ No longer can be selected

☐ Show the Center points

- **No longer can be selected** - This option enables you to select drills that were not selected previously.
- **Show the Center points** - This option enables you to see the drills that were picked in the previous drill geometry.

4.4 3D Model Geometry

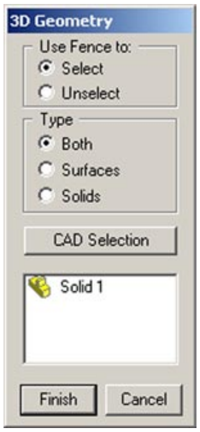
Any surface, solid or a combination of surfaces and solids can be selected as a 3D geometry. These models can be machined with the **3D Model Job** in SolidCAM using various milling strategies.



4.4.1 Defining a 3D Model Geometry

For 3D milling you need to build or import a model representation of your workpiece or part in the CAD system. This model describes the actual part with all the dimensions and topology information. By adding a 3D model to the CAM-Part, you enable SolidCAM to use this model to calculate the CNC tool path.

1. After you select the commands **Add>3D Model**, the **3D Geometry dialog** will be displayed.



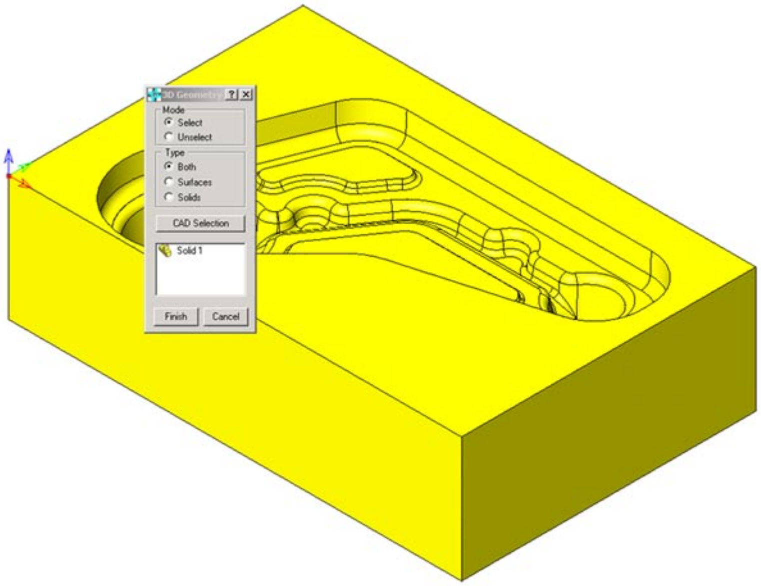
3D Geometry
Use Fence to:
● Select
○ Unselect
Type
● Both
○ Surfaces
○ Solids
Solid 1

2. Choose the selection mode in the **Type** field. You can choose the following options:

    - **Solid** - to select only solid objects;
    - **Surfaces** - to select surfaces only;
    - **Both** - to select both surfaces and solids.

The default option is **Both**.

3. Select the solid body.



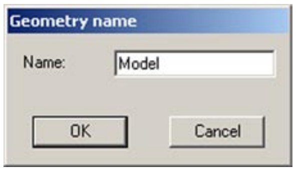
Mode
● Select
○ Unselect

Type
● Both
○ Surfaces
○ Solids

Solid 1

4. The selected entities will be highlighted and the selected entity name will be displayed. In case you have chosen wrong entities, use the **Unselect** option to undo your selection. You can also right click on the entity name (the object will be highlighted) and choose the **Unselect** option from the menu. Confirm your selection by clicking on the **Finish** button.

5. Confirm the **Default Geometry Name** (see the topic **10.12.**) or enter a new name for the geometry into the **Geometry name dialog**.

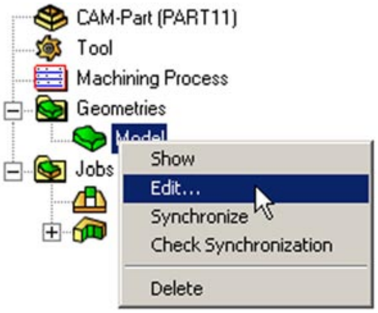


Name: Model

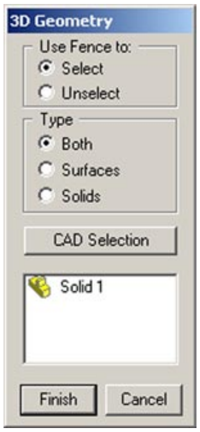
4.4.2 Editing a 3D Model Geometry

In the CAM module we refer to edit as adding, removing or replacing entities in 3D geometries that have been previously saved. Actual changes in dimensions or topology of your model must be performed in the CAD module.

1. In the **CAM Manager**, right click on the **Geometry** field. Select **Edit** from the menu.


CAM-Part (PART11)
Tool
Machining Process
Geometries
Model
Jobs
Show
Edit...
Synchronize
Check Synchronization
Delete

2. The **3D Geometry dialog** will be displayed.

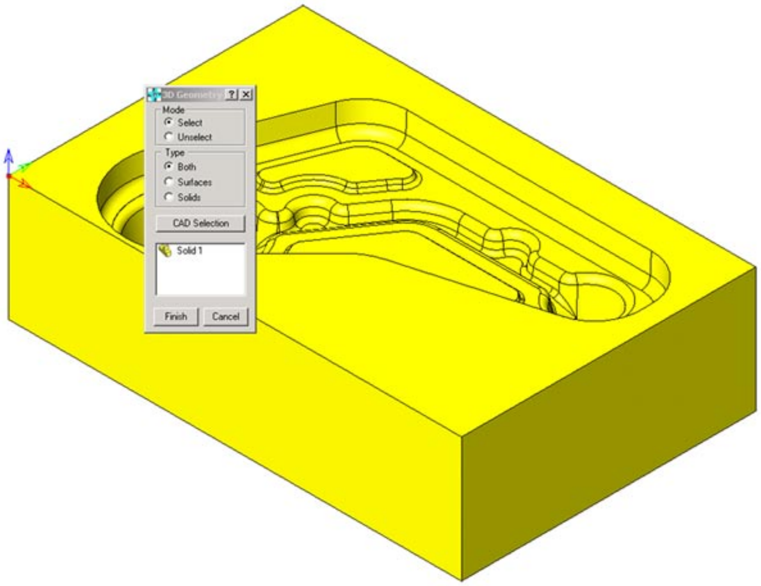

3D Geometry

Use Fence to:
● Select
○ Unselect

Type
● Both
○ Surfaces
○ Solids

CAD Selection
Solid 1
Finish
Cancel

3. The default selection will add solids and surfaces to your model. To remove or replace solids or surfaces from your model, use the **Unselect** option. Click on the entities you want to add/remove from your 3D model geometry.


3D Geometry
Mode
● Select
○ Unselect
Type
● Both
○ Surfaces
○ Solids
CAD Selection
Solid 1
Finish
Cancel

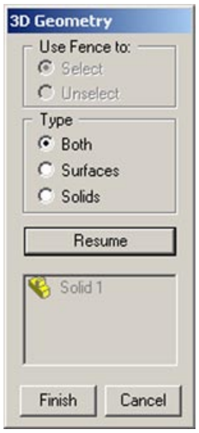
4. In case you want to replace a model, activate the **Select** option and add the updated model to your geometry.

5. Confirm your selection by clicking on the **Finish** button.

You can use the **CAD selection** mode to select the geometry with host CAD tools.

4.4.3 CAD selection

This option enables you to select the 3D geometry with the host CAD tools.


3D Geometry
Use Fence to:
● Select
○ Unselect
Type
● Both
○ Surfaces
○ Solids
Resume
Solid 1
Finish
Cancel

1. Click on the **CAD Selection** button in the **3D Geometry dialog**. The 3D Geometry dialog will be changed:

2. Select solid or surface objects (depending on the option chosen in the **Type** field.

3. Click on the **Resume** button to finish the selection.

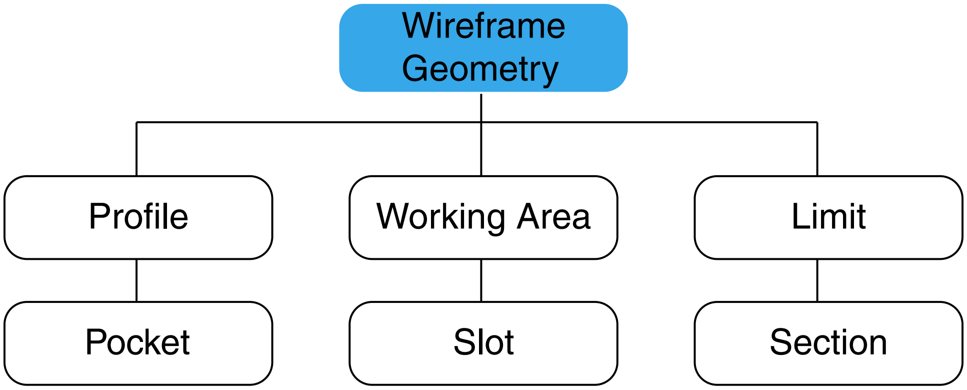
4. Confirm your selection by clicking on the **Finish** button.

5. Enter a name for the geometry into the **Geometry name dialog**.

4.5 Wireframe Geometry

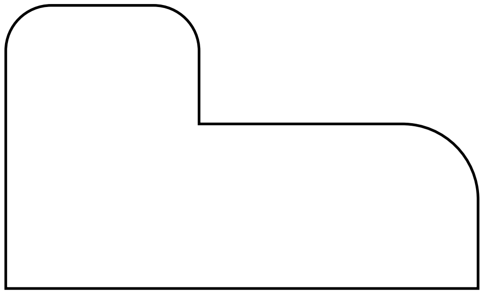
Wireframe geometry has sub-types; each with its own set of rules. All the sub-types use the same interface to select the geometry. In this lesson, the common selection methods will be explained without the special rules relevant for each sub-type.

Chain geometries are defined from the following entities: edges of models, 2D curves, 3D curves, circles, lines and splines. Each chain is composed from one or more entities and defines an open or closed contour.

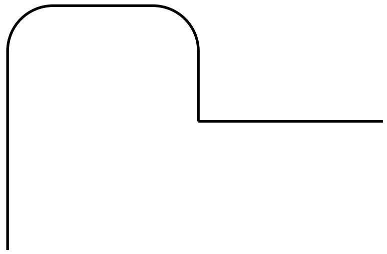

Wireframe Geometry
Profile
Working Area
Limit
Pocket
Slot
Section

4.5.1 Defining a Profile/Pocket Geometry

Chains for **Profile Milling** (see the topic **6.1**.) can be either open or closed. You can machine one or more profiles in one operation. In case you want to add more chains to the geometry, select the **Add chain** command again.

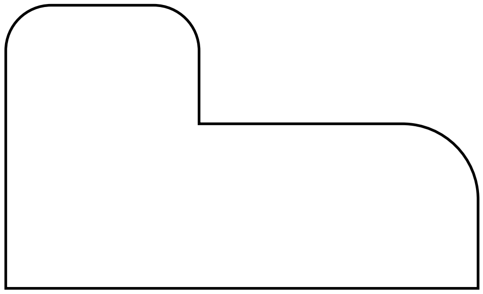


Closed chain

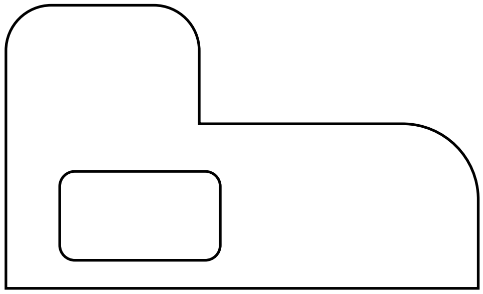


Open chain

Chains for **Pocket Milling** (see the topic **6.2**.) must be closed. The first chain defines the contour of the pocket. All closed chains inside the first chain of each pocket will automatically be treated as **pocket islands**. Overlapping chains will be milled as separate pockets - not as islands. To select multiple pockets with islands, continue adding chains to the geometry.

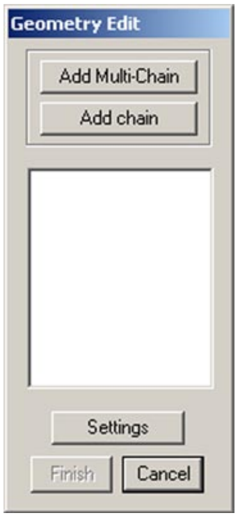


Closed chain



Closed chain with island

1. After you have selected the command **Add Profile**, the **Geometry Edit dialog** will be displayed.
2. Choose the **Add chain** button or the **Multi-chain** command to define the chains.
3. The **Chain options dialog** (see the topic **4.7.1**.) or the **Chain Selection dialog** (see the topic **4.7.2**.) will be displayed.
4. When you have completed the geometry definition, confirm the **Geometry Edit dialog** (see the topic **4.6**.) with **Finish**.
5. Confirm the **Default Geometry Name** or enter a new name for the geometry into the **Geometry name dialog** and confirm with **OK**.

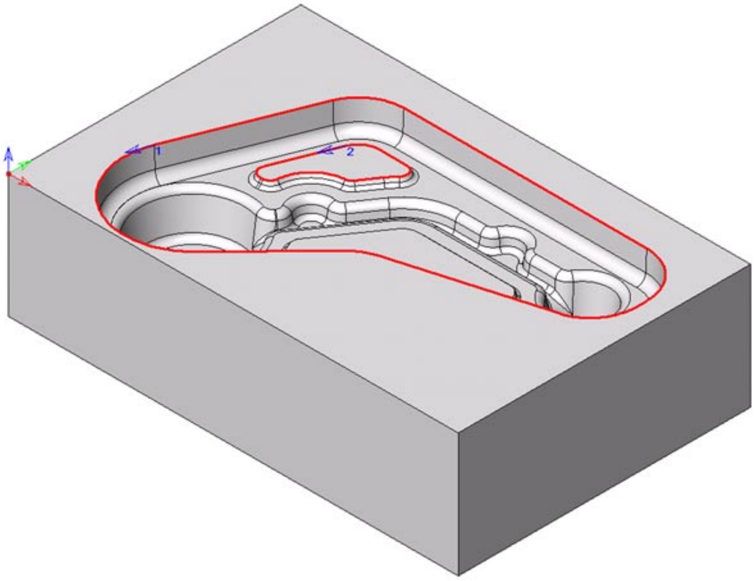


4.5.2 Defining a Working Area

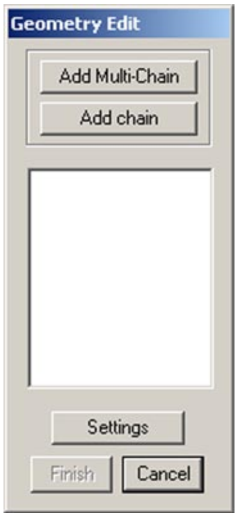
Working areas are used in **3D milling** (see the topic **7.2**.) to constrain the machining to certain areas of the 3D model. A working area is defined by a closed chain. The chain can be either selected from points or from any 2D or 3D edges, curves or splines.

As in pocket geometries, working areas can be defined with internal chains, so that the islands will be excluded from machining.

In this **Example** the working area has been selected to re-work the radius along the cavity. Chain1 defines the outside boundary of the area; chain 2 excludes the cavity from machining.

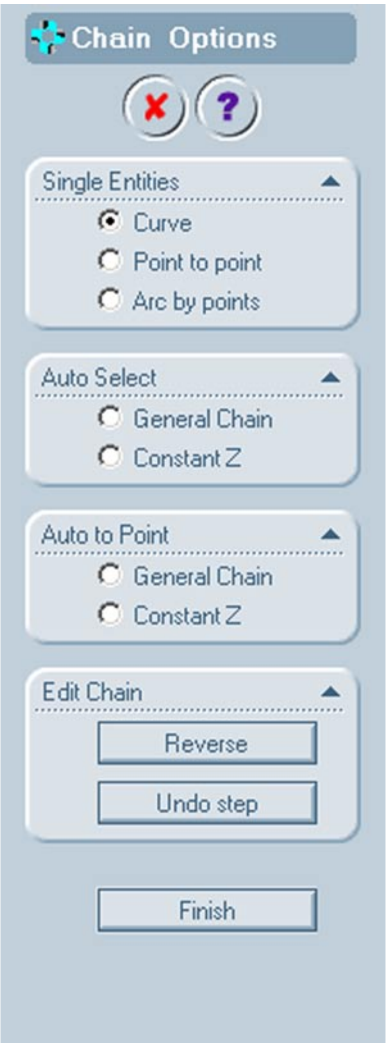


1. After you have selected the command **Add>Working area**, the **Chain Geometry dialog** is displayed.



2. Click on the **Add chain** button or choose the **Multi-chain** mode.

3. The **Chain options dialog** is displayed.


Chain Options
Single Entities
● Curve
○ Point to point
○ Arc by points
Auto Select
○ General Chain
○ Constant Z
Auto to Point
○ General Chain
○ Constant Z
Edit Chain
Reverse
Undo step
Finish

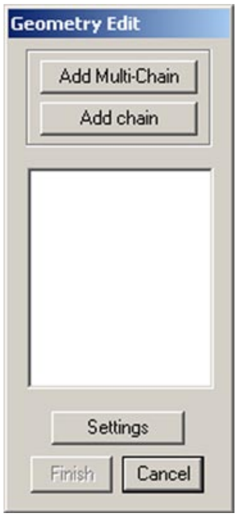
4. When you have completed the geometry definition, confirm the **Chain Geometry** dialog with **Finish**. In case you want to define another working area or an island inside the previous chain, repeat the last steps.

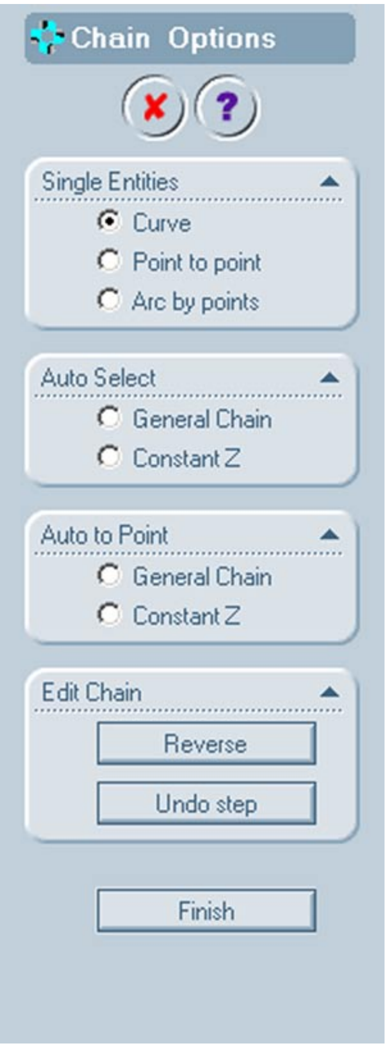
5. Confirm the **Default Geometry Name** or enter a new name for the geometry into the **Geometry name dialog** and confirm with **OK**.

4.5.3 Defining a Slot Geometry

Chains for **Slot Milling** (see the topic **6.4**.) can be either open or closed. You can machine one or more slots in one operation. If you want to add more chains to the geometry, select the Add chain command again.

1. After you have selected the command **Add>Slot**, the **Chain Geometry dialog** is displayed.

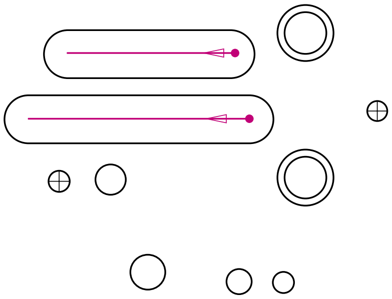

Geometry Edit
Add Multi-Chain
Add chain
Settings
Finish
Cancel


Chain Options
Single Entities
● Curve
○ Point to point
○ Arc by points
Auto Select
○ General Chain
○ Constant Z
Auto to Point
○ General Chain
○ Constant Z
Edit Chain
Reverse
Undo step
Finish

2. Click on the **Add chain** button or choose the **Multi-chain** mode.

3. The **Chain options dialog** is displayed.

4. Click on the edges or lines that make up your **Slot** and confirm with **Finish**.



5. When you have completed the geometry definition, confirm the **Chain Geometry dialog** with **Finish**.

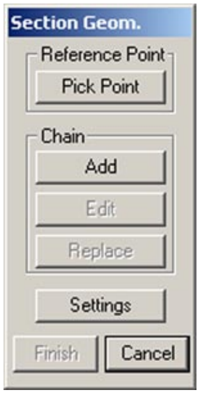
6. Confirm the **Default Geometry Name** or enter a new name for the geometry into the **Geometry name dialog** and confirm with **OK**.

4.5.4 Defining a Section Geometry

In the **Translated Surface Job** (see the topic **6.5**.) and **Slot Job** (see the topic **6.4**.) you need to select a section geometry. These 2.5D jobs can be defined using 2D geometries only. With section geometries you have to define depth profiles for Slot and Translated surface jobs.

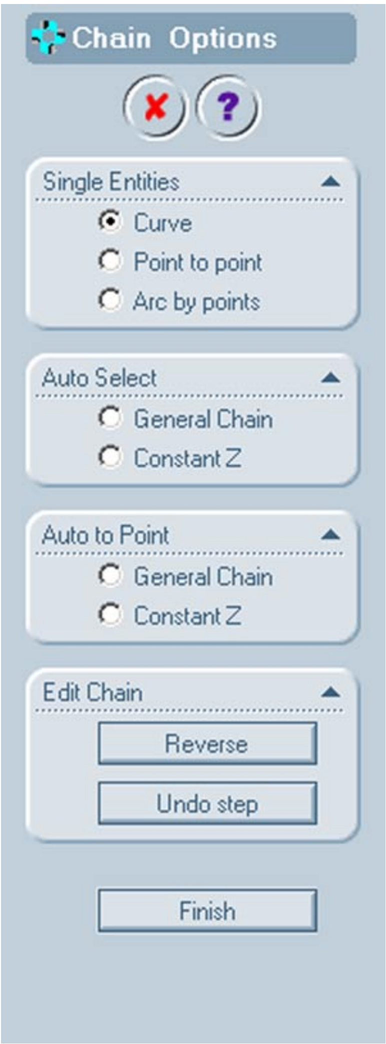
Add a Section Geometry

1. After you have selected the command **Add>Section**, the **Section Geometry dialog** is displayed.

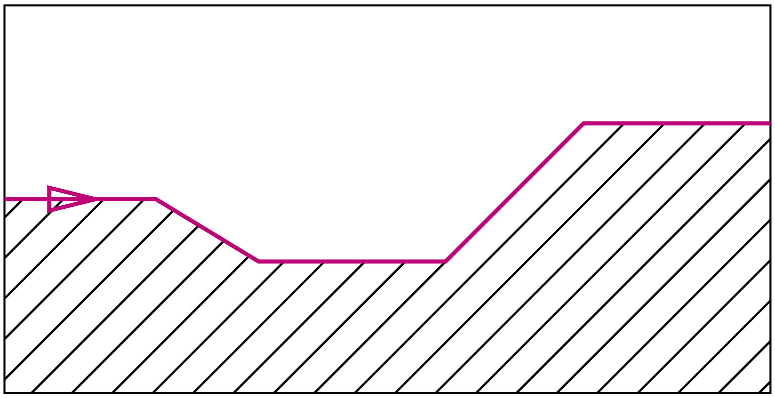

Section Geom.
Reference Point
Pick Point
Chain
Add
Edit
Replace
Settings
Finish
Cancel

2. Click on the **Add chain** button.

The **Chain options dialog** is displayed.


Chain Options
Single Entities
● Curve
○ Point to point
○ Arc by points
Auto Select
○ General Chain
○ Constant Z
Auto to Point
○ General Chain
○ Constant Z
Edit Chain
Reverse
Undo step
Finish

3. Click the edges or lines that make up your **Section** and confirm with **Finish**.



4. Click on the **Point** button to pick the **Reference point**. This point defines the level of the section relative to the upper job plane.

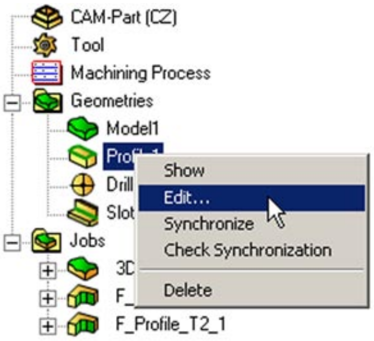
5. Pick this point using the CAD point selection options. Confirm the **Section Geometry dialog** with **Finish**.

6. Confirm the **Default Geometry Name** or enter a new name for the geometry into the **Geometry name dialog** and confirm with **OK**.

Edit a Section Geometry

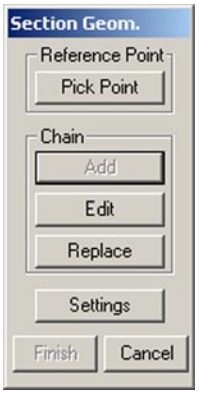
You can edit the section or re-define the reference point of the section.

1. In the **CAM Manager**, right click on a Section geometry and select Edit from the menu.


CAM-Part (CZ)
Tool
Machining Process
Geometries
Model1
Profi
Drill
Slot
Jobs
3D
F_
F_Profile_T2_1
Show
Edit...
Synchronize
Check Synchronization
Delete

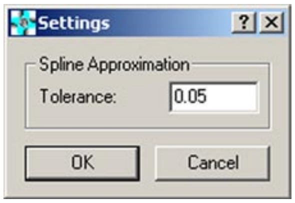
2. In the **Section Edit dialog**, click on **Point** to re-pick the Reference point or **Edit/Replace** the section chain using the **Chain options dialog**.

3. Use **Finish** to confirm your changes and close the dialog.


Section Geom.
Reference Point
Pick Point
Chain
Add
Edit
Replace
Settings
Finish
Cancel

Settings

The **Settings dialog** will be displayed.


Settings
Spline Approximation
OK
Cancel

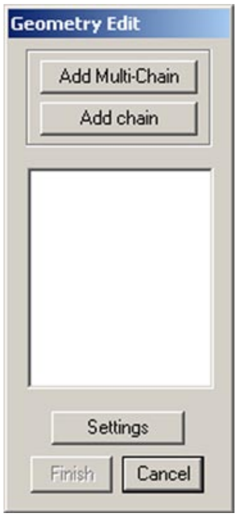
Tolerance: 0.05

This dialog enables you to define the **Spline approximation tolerance** for the chain selection.

4.5.5 Defining a Limit Geometry

A **Translated surface job** (see the topic **6.5**.) can be defined using 2D geometries only. With **Limit** geometries you can define boundaries for these surface jobs.

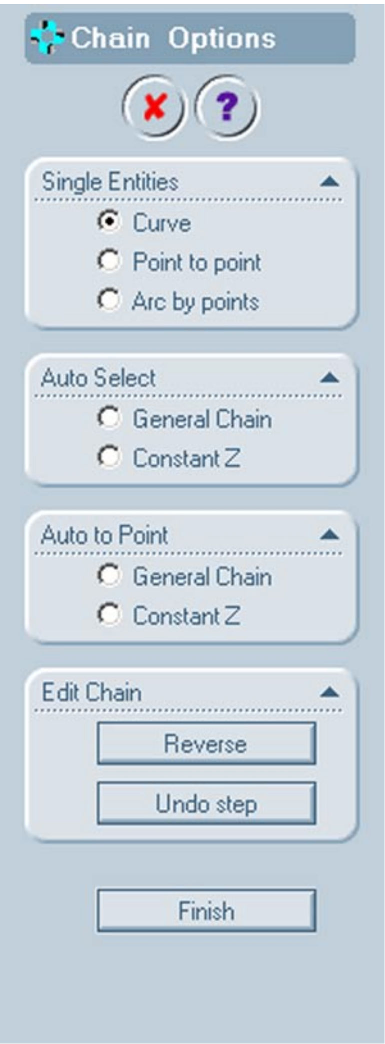
1. After you have selected the command **Add Limit**, the **Chain Geometry dialog** is displayed.

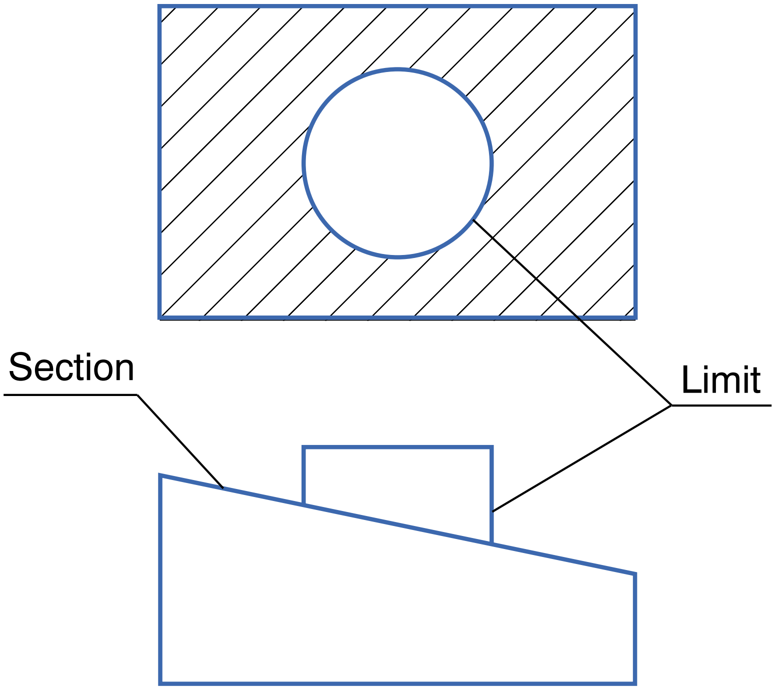

Geometry Edit
Add Multi-Chain
Add chain
Settings
Finish
Cancel

2. Click on the **Add chain** button or choose the **Multi-chain** mode.

3. The **Chain options dialog** is displayed.

4. Click on the edges or lines that make up your **Limit** and confirm with **Finish**.


Chain Options
Single Entities
● Curve
○ Point to point
○ Arc by points
Auto Select
○ General Chain
○ Constant Z
Auto to Point
○ General Chain
○ Constant Z
Edit Chain
Reverse
Undo step
Finish

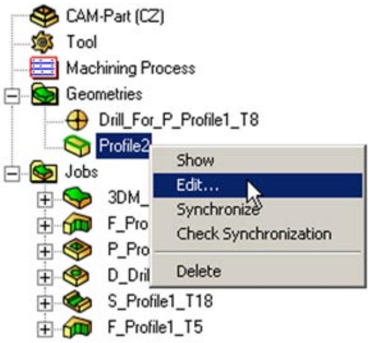


5. When you have completed the geometry definition, confirm the **Chain Geometry** dialog with **Finish**.

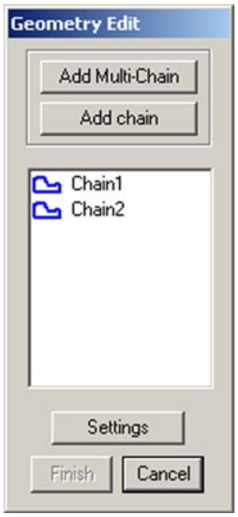
6. Confirm the **Default Geometry Name** or enter a new name for the geometry into the **Geometry name dialog** and confirm with **OK**.

4.6 Editing Chain Geometries

To edit a chain, press the right mouse button on the **Geometry** field and select **Edit** from the sub-menu in the **CAM Manager**.


CAM-Part (CZ)
Tool
Machining Process
Geometries
Drill_For_P_Profile1_T8
Profile2
Jobs
3DM_
F_Pro
P_Pro
D_Dril
S_Profile1_T18
F_Profile1_T5
Show
Edit...
Synchronize
Check Synchronization
Delete

The **Chain geometry dialog** will be displayed.


Geometry Edit
Add Multi-Chain
Add chain
Chain1
Chain2
Settings
Finish
Cancel

Select a command from the menu.

4.6.1 Add Multi-chain

This option enables you to create a number of chains from a model. The program will automatically create chains from the elements that are selected.

For more information on this subject, please refer to topic **4.7.1**.

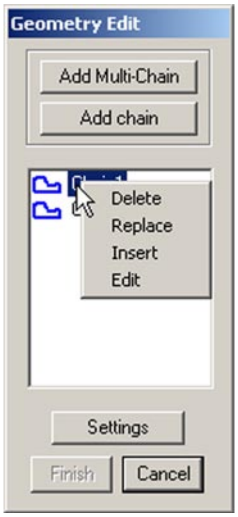
4.6.2 Adding a Chain

You can add a chain to the existing set of chains in the current geometry. The new chain will be added under the next sequential number. Define the new chain using the **Chain options dialog**.

For more information on this subject, please refer to topic **4.7.2**.

4.6.3 Chain operations

All the selected chains are displayed in this dialog. To edit these chains, right click on the Chain name in the **Geometry edit dialog** and choose the appropriate command from the menu.


Geometry Edit
Add Multi-Chain
Add chain
Delete
Replace
Insert
Edit
Settings
Finish
Cancel

Deleting a Chain

Deletes a chain from the current geometry.

Replacing a Chain

This command enables you to update a chain in the current chain geometry. The old chain will be deleted and you can now define the new chain by using the **Chain options dialog**.

Inserting a Chain

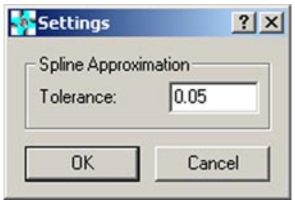
This command enables you to insert a chain between two existing chains. Define the new chain using the **Chain options dialog**.

Editing a Chain

This option enables you to edit an existing chain. The **Chain options dialog** will be displayed. You can reverse the chain or undo the selection steps to change the chain.

4.6.4 Settings

The **Settings dialog** will be displayed.



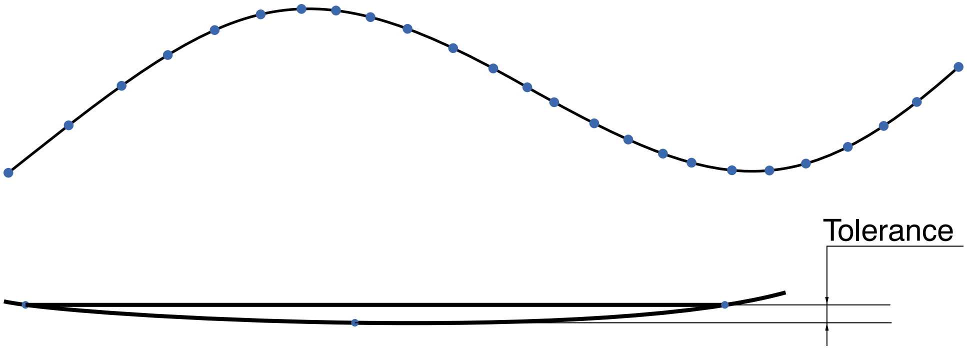
Tolerance: 0.05

This dialog enables you to define the **Spline approximation tolerance** for the chain selection.

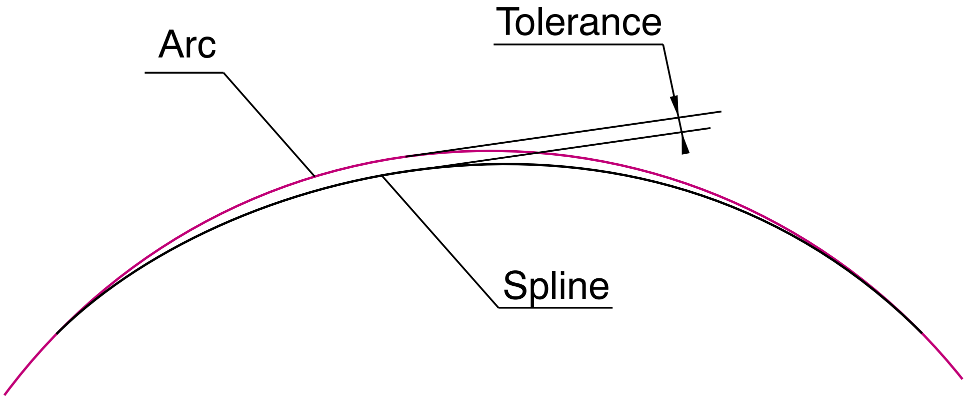
4.6.5 Spline approximation

SolidCAM enables you to define the **Spline approximation** tolerance for the chain selection. Arcs on the XY plane are saved as arcs in the SolidCAM database; if you select an arc or spline that is not on the XY plane of the actual part home position, SolidCAM will do the following:

1. Project the arc or spline on the XY plane of the active home and divide it into segments according to the specified tolerance

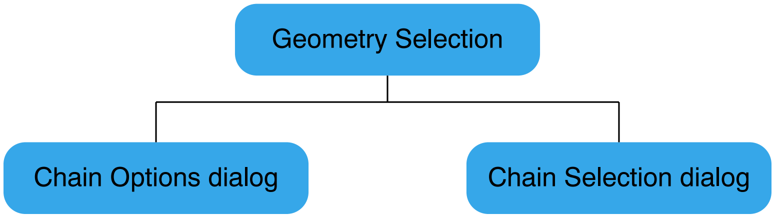


2. In order to reduce the number of segments, SolidCAM tries to fit arcs on the segments generated in step 1 according to the specified tolerance multiplied by 3.



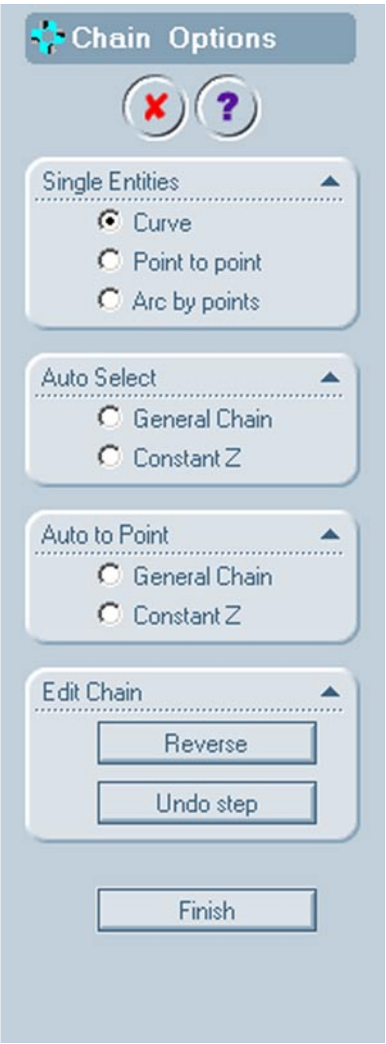
4.7 Chain selection

SolidCAM enables you to choose between two different tools to select a geometry:


Geometry Selection
Chain Options dialog
Chain Selection dialog

4.7.1 Chain Options dialog

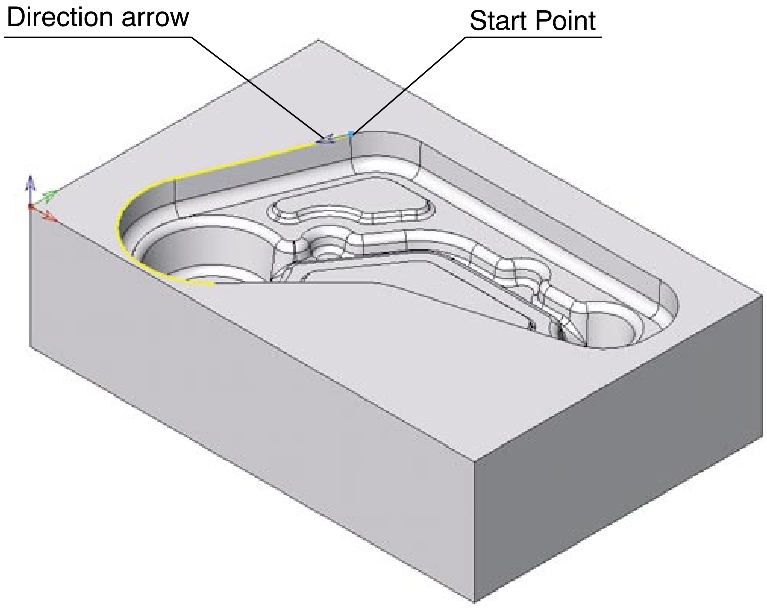
The chain selection options in SolidCAM enable you to select contours and boundaries quickly and easily. Whenever you have to define a chain geometry, the **Chain options dialog** is displayed. This dialog is built in the SolidWorks PropertyManager.


Chain Options
Single Entities
● Curve
○ Point to point
○ Arc by points
Auto Select
○ General Chain
○ Constant Z
Auto to Point
○ General Chain
○ Constant Z
Edit Chain
Reverse
Undo step
Finish

You can switch between the different chain options when defining a chain. The default option is the **Curve** option in the **Single Entities** section; you can, however, start the chain definition with any other option.

Chain Direction

Some of the jobs in SolidCAM use the direction of the chain geometry to calculate the tool path. In Profile Milling, for example, you have to specify the tool side, which is relative to the direction of the selected contour. The edges or sketch segments that you selected will be highlighted. The arrow at the start point of the chain indicates the direction of the chain.



Use the **Reverse** command to reverse the direction of the chain - during or after the chain selection, if necessary.

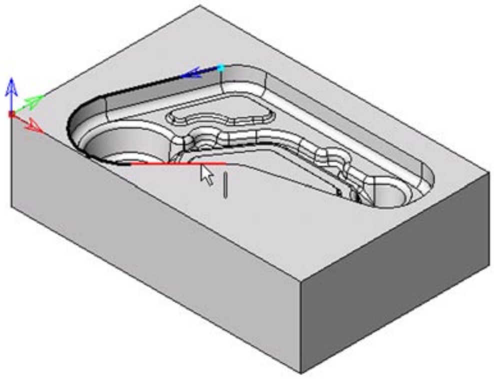
Single Entities

You can define the contour by selecting edges, sketch segments and points on the contour. SolidCAM provides two options:

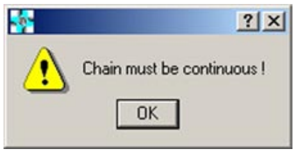
- **Curves**

  You can create a chain of existing curves and edges by selecting them one after the other.

  1. **Pick start curve** - Select the start entity (a wireframe element or the solid model edge).



2. **Pick next curve** - Select the next curve that belongs to the chain. The curve must be connected to the previous curve; otherwise, the following message will be displayed:



Chain must be continuous !

3. SolidCAM enables you to define the **tolerances** to determine whether two consequent elements should be connected or not.

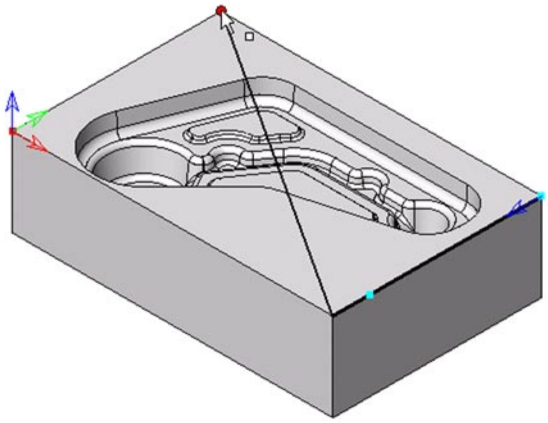
4. To complete the selection, select **Finish** from the **Chain option dialog**.

Associativity

SolidCAM will keep the associativity to any edge or sketch entity. Any change made to the model or sketch will automatically update the selected geometry.

- **Point to point**

This option enables you to connect specified points; the points will be connected by a straight line.



1. **Pick start point** - Either enter the coordinates into the command line or pick a point.

2. **Pick next point** - A chain element will be created from the previous point up to this point. Either enter the coordinates into the command line or pick a point.

3. To complete the selection, select **Finish** from the **Chain option dialog**.

Associativity

SolidCAM will not keep the associativity to any selected point. SolidCAM saves the X, Y and Z coordinates of the selected points. Any change made to the model or sketch will not update the selected geometry.

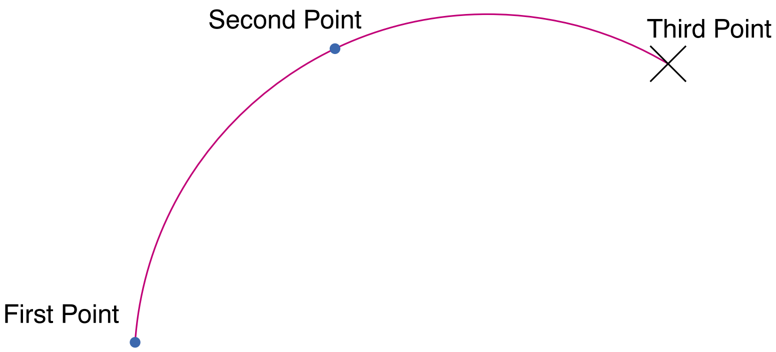
You cannot select a point that is not on a SolidWorks entity (if you need to select such a point, add a planer surface under the model and select the points on that surface).

The following rules have to be understood when using virtual line selection by the **Point to Point** option:

1. When you select a virtual line between two edges, the line will behave as a spring. Whenever the model is changed and synchronized, the geometry will be updated with the model.
2. When you select a sequence of more than one virtual line, only the points connected to model edges or sketch elements will be updated, but all the other points will stay fixed at the defined X, Y and Z positions.

- **Arc by points**

This option enables you to create a chain segment on an arc upto a specific point on the arc.



1. **Pick start point** - Choose the point on the arc where you want to start. If this is not the first point on the chain, go to step **2**.
2. **Pick next point** - Choose a point on the arc that is between the first point on the arc and the third point on the arc.
3. **Pick end point** - Choose the third point on the arc where you want the chain to end.

Associativity

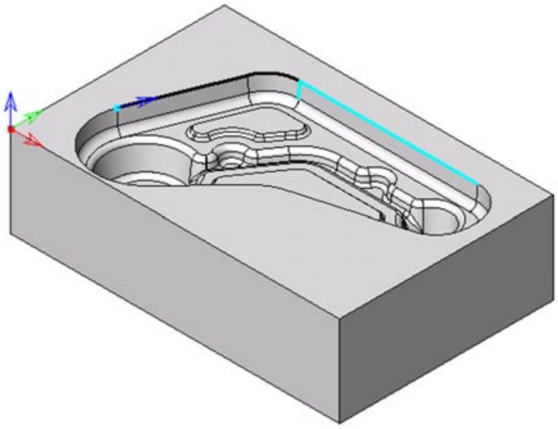
SolidCAM will not keep the associativity to any selected arcs by points. SolidCAM saves the X, Y and Z coordinates of the selected points. Any change made to the model or sketch will not update the selected geometry.

Auto Select

SolidCAM will automatically find the chain entities and try to close the chain contour. The Auto select mode offers the following three different options:

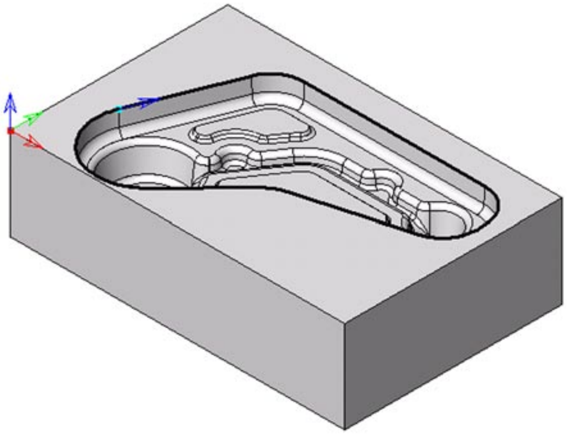
- **General chain**

  SolidCAM will **highlight all entities** that are connected to the last chain entity. You will have to select the entity along which you want the chain to continue.



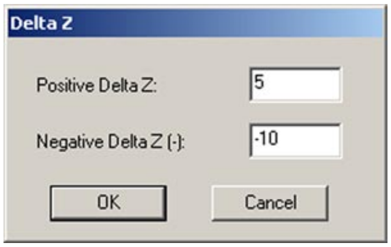
- **Constant-Z**

  This option will identify only entities on the same XY-plane with the previously selected chain entity. You will be only prompted to identify the next chain element when two entities on the same Z-level are connected to the chain. The system tolerance for this option can be set in the **CAM settings**.



- **Delta-Z**

When you select this option, you will be required to enter a positive and negative Z deviation into the **Delta-Z dialog**. Only entities inside this range will be identified as the next possible entity of the chain.



Positive Delta Z: 5

Negative Delta Z (-): -10

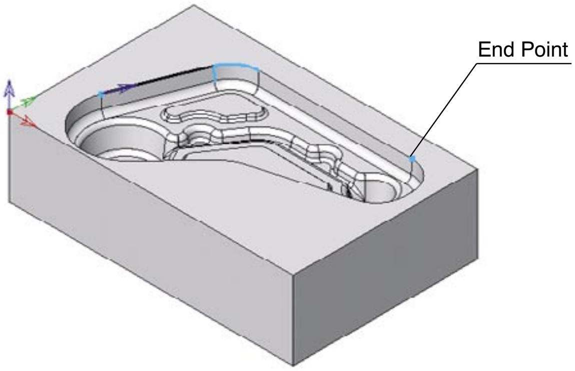
1. **Pick start curve** - Select the start entity of the chain.
2. **Pick point in chain direction** - Indicate the chain direction by choosing a point on the selected start entity of the chain.
3. You will be prompted to **Select one of the possible curves** if the system detects that more than one entity could be the next element of the chain; indicate your choice.
4. When the system can close the chain, you are asked whether it is **OK to accept**; choose **Yes** to accept. If you answer **No**, you can edit the chain selection in the Chain options menu. You can, for example, undo the last step or cancel the chain selection.

Auto to Point

The chain is selected by specifying the start curve, the direction of the chain and the vertex upto which the chain will be created. This command is useful if you do not want to define a closed chain, but an open chain upto a certain point. The Auto to Point selection offers three different options:

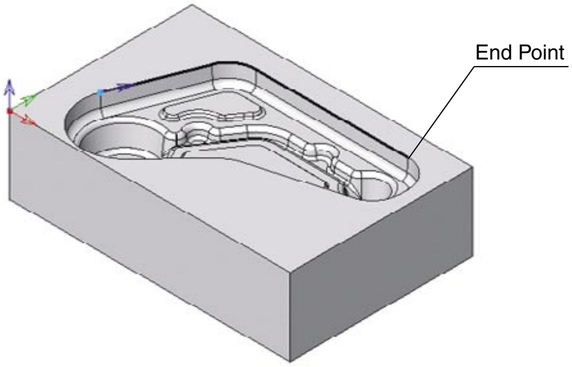
- **General chain**

SolidCAM will highlight all entities that are connected to the last chain entity. You will have to select the entity along which you want the chain to continue.



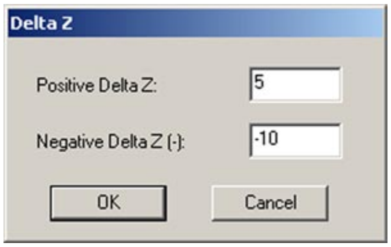
- **Constant-Z**

This option will identify only entities on the same XY-plane with the previously selected chain entity. You will be only prompted to identify the next chain element when two entities on the same Z-level are connected to the chain. The system tolerance for this option can be set in the **CAM settings**.



- **Delta-Z**

When you select this option, you will be required to enter a positive and negative Z deviation into the **Delta-Z dialog**. Only entities inside this range will be identified as the next possible entity of the chain.



Positive Delta Z: 5

Negative Delta Z (-): -10

1. **Pick start curve** - Select the start entity of the chain.
2. **Pick point in chain direction** - Indicate the chain direction by picking a point on the selected start curve of the chain.
3. **Pick target vertex on chain** - Select the end point of the end entity of the chain.
4. You will be prompted to **Pick one of the possible curves** if the system detects that more than one entity could be the next element of the chain; indicate your choice.
5. Confirm your chain with **Finish** in the **Chain option dialog**.

Edit Chain

- **Reverse**

  This option enables you to **reverse** the direction of the chain you are currently working with. The direction is indicated by an arrow at the chain's start point.

- **Undo step**

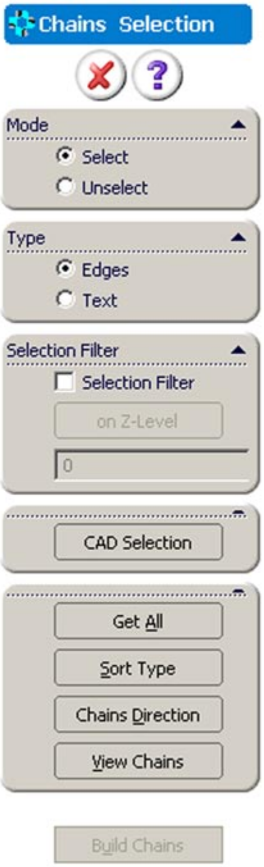
  This option enables you to undo the last selection of a chain element.

Finish

Confirms the selection of the chain and proceeds to the next step.

4.7.2 Chain selection dialog

The **Chain selection dialog** enables you to choose a number of chains from a model by selecting the model elements. The program will automatically create chains from the selected elements.


Chains Selection
Mode
Type
Selection Filter
on Z-Level
0
CAD Selection
Get All
Sort Type
Chains Direction
View Chains
Build Chains

Mode: ● Select ○ Unselect

Type: ● Edges ○ Text

☐ Selection Filter

Mode:

- **Select** - This mode enables you to select the elements to define the chains.
- **Unselect** - This mode enables you to unselect elements that were previously selected.

Type

- **Edges**

  When this field is activated, SolidCAM enables you to choose model edges to build a chain geometry.

- **Text**

  When this field is activated, SolidCAM enables you to choose text to build a chain geometry.

Selection Filter

When this field is activated, it enables you to select only the elements that are on a specific **Z level**. You can enter the **Z level** value in two ways: either enter the value in the dialog window or press the **On Z-Level** button.

- **On Z-Level** - Pressing this button will enable you to choose the **Z level** directly from the model.

CAD selection

With this button, SolidCAM enables you to select a geometry for chains with the Host CAD tools.

Get All

This button enables you to choose all the elements.

Sort Types

When there is more than one chain, this option will enable you to determine the chain you would like to start from. The **Sort chains dialog** will be displayed (see the topic **4.7.4**).

Chains Direction

This option enables you to determine the cutting direction of each chain. The **Chains direction dialog** will be displayed (see the topic **4.7.3**).

View Chains

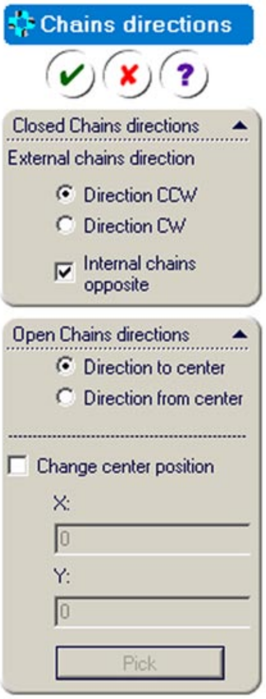
The **View Chains** option enables you to see the chains before they are created.

Build Chains

After you select the chains you need, you can use this option to create the chains.

4.7.3 Chains direction

The **Chain direction** option enables you to determine the direction of the chain.



Chains directions

Closed Chains directions

External chains direction

● Direction CCW

○ Direction CW

☑ Internal chains opposite

Open Chains directions

● Direction to center

○ Direction from center

☐ Change center position

X: 0

Y: 0

Select open chain with:

This option is used in cases where the chain being used is an open chain.

- **Direction to center**: The chain start point will be the end of the chain furthest from the given center point.
- **Direction from center**: The chain start point will be the end of the chain closest to the given center point.

Change center position:

Tagging this will enable you to choose the center position either by entering an X and Y value or by picking a point on the model.

If you do not change the center position, the home position will be used as the center point.

Select closed chains:

This option enables you to determine the direction of closed chains.

External chains with:

External chains are the outer borders of a pocket. You can determine if you want the chain to go clockwise or counterclockwise by choosing **Direction CCW** or **Direction CW**.

Internal chains opposite:

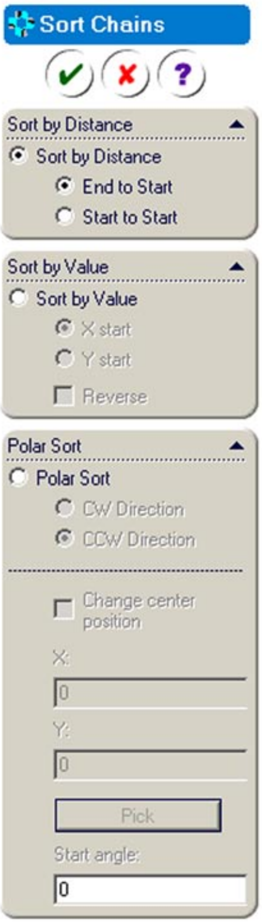
Internal chains are within the boundaries of external chains. By tagging this option you can have the internal chains go in the opposite direction of the external chains.

4.7.4 Chains sorting

Chain sorting enables you to decide the order in which you want the chains to be created.

By Distance

The chain that is closest to the last chain will be chosen using two possible options.



Sort by Distance
● Sort by Distance
● End to Start
○ Start to Start

Sort by Value
○ Sort by Value
● X start
○ Y start
☐ Reverse

Polar Sort
○ Polar Sort
○ CW Direction
● CCW Direction
☐ Change center position
X: 0
Y: 0

Start angle: 0

- **From End - Start**

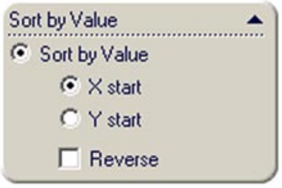
  The next chain will be the chain with the closest start point from the end point of the last chain.

- **Start - Start**

  The next chain will be the chain with the closest start point from the start point of the last chain.

By Value

The next chain that is closest to the last chain in a given direction will be chosen.


Sort by Value
● Sort by Value
● X start
○ Y start
☐ Reverse

- **X start**

  The next chain will be the one in the X direction.

- **Y start**

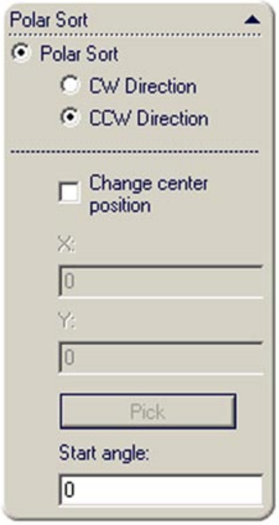
  The next chain will be the one in the Y direction.

- **Reverse**

  Reverse changes the direction the chains. (X-start: Left to Right, Y-start: Upward or Downward).

Polar

Chains will be chosen around a specific point. If no point is chosen, the default point is the home point of the CAM-Part.


Polar Sort
● Polar Sort
○ CW Direction
● CCW Direction
☐ Change center position
X:
0
Y:
0
Pick
Start angle:
0

- **CW Direction**

  The order of the chains will be in clockwise direction around the point.

- **CCW Direction**

  The order of chains will be in counterclockwise direction around the point.

- **Change center position**

  The center position will be chosen either by entering X/Y values or by picking a point on the model.

- **Start angle**

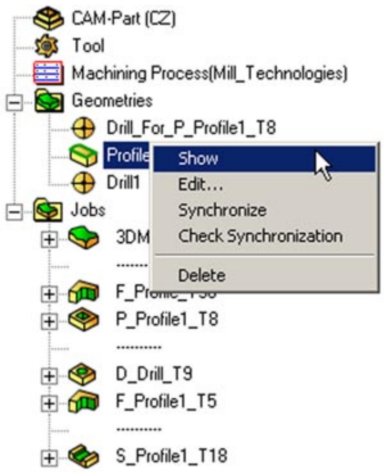
  This controls the angle between the start point of the chain and the point of rotation.

4.8 Operations with Geometries

You can edit and manage your geometries from the Geometry field of the CAM-Manager. Geometries that are used by machining jobs can be edited through the CAM-Manager as they appear under the corresponding job. SolidCAM also enables you to define and show geometries inside the job menu, so that you can select the geometry and machining strategy in one step.

1. In the CAM Manager, move the mouse on a **Geometry** field and press the **right** button.

The **Geometry menu** will be displayed.

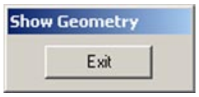

CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Drill_For_P_Profile1_T8
Profile
Drill1
Jobs
3DM
F_Pro
P_Profile1_T8
D_Drill_T9
F_Profile1_T5
S_Profile1_T18
Show
Edit...
Synchronize
Check Synchronization
Delete

2. Select the command from the menu.

Show

With the Show command, the 3D model, drill or chain geometry will be displayed in the active CAD window.

To exit from this mode, use the Exit button in the displayed **Show Geometry dialog**.


Show Geometry
Exit

Edit

You can change or update the selected geometry. If the tool path of a job has been calculated, it will be deleted. You have to re-calculate the job after you make changes to the geometry.

Delete/Delete All

You can delete geometries that are not in use by any machining operation (job). These geometries are displayed under the Geometries symbol. Any geometry in use by jobs will appear under the respective job(s) only and cannot be deleted unless the job is deleted as well.

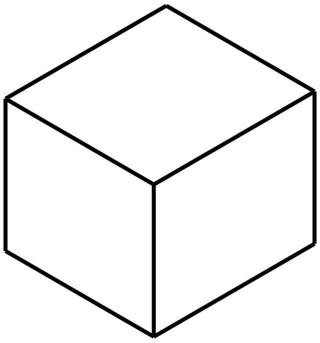
The **Delete all** command is only available when you **right** click on the main topic symbol **Geometries**. It will delete all geometries not currently used by any machining operation. Whenever you click on a single geometry, the **Delete** command will be active for the selected geometry only.

Synchronize/Check synchronization

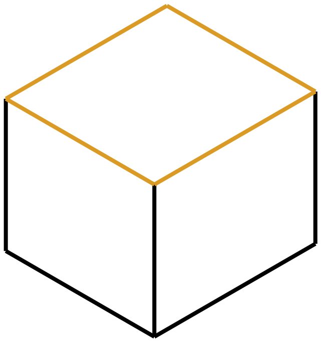
When the SolidWorks model is updated; SolidCAM enables you to check the compliance of the SolidCAM geometry to the updated SolidWorks model. If the SolidCAM geometry is not synchronized with the SolidWorks model, SolidCAM enables you to synchronize it.

4.8.1 Synchronization of the Wireframe geometry

SolidCAM Wireframe geometry is based on the SolidWorks model elements like edges, vertexes or sketches.

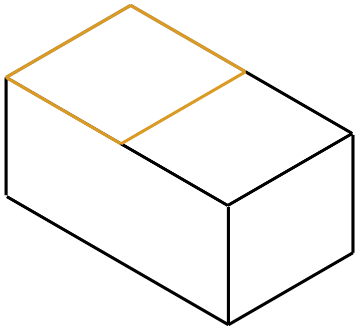


SolidWorks Model

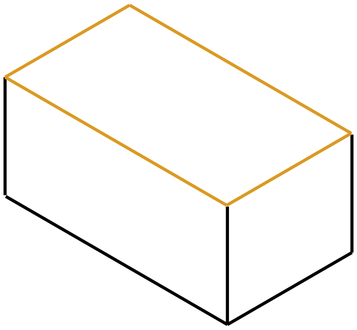


SolidCAM Geometry
based on the
SolidWorks Model

When the SolidWorks model is changed, SolidCAM uses the **synchronization algorithm** to synchronize the SolidCAM geometry with the updated SolidWorks model.



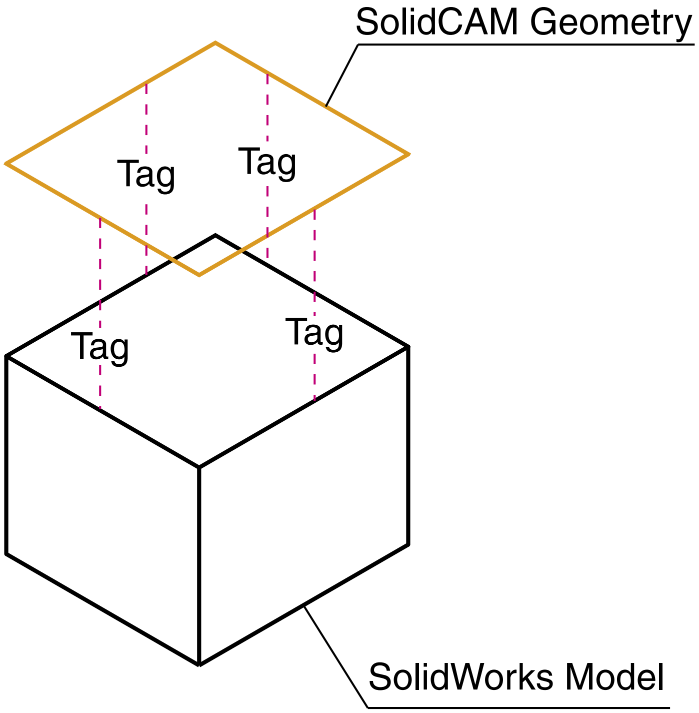
SolidWorks Model is changed, SolidCAM Geometry have to be updated



SolidCAM Geometry is updated

Synchronization check

SolidCAM marks all the SolidWorks entities used for the geometry definition with a tag. The list of these tags is saved. The tags enable SolidCAM to establish a connection between the SolidCAM geometry and the SolidWorks model entities. The relations between the tagged SolidWorks geometry and SolidCAm geometry is saved in the geometry file ***.gem**.

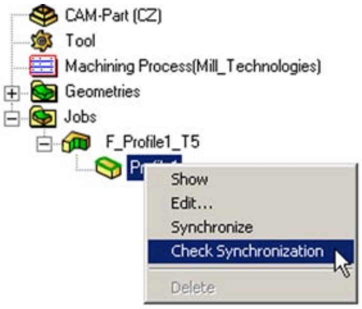


SolidCAM enables you to check the synchronization status automatically or manually depending on the **Synchronization settings** (see the topic **10.11**.).

When you need to check the synchronization status of the geometry manually, do the following steps:

1. Right click on the Job or Geometry name in the SolidCAM Manager.

2. Choose the **Check Synchronization** item from the menu.

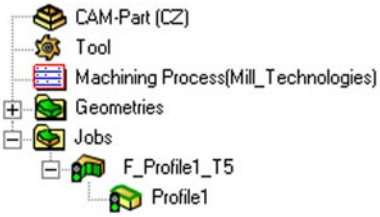

CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
F_Profile1_T5
Profile1
Show
Edit...
Synchronize
Check Synchronization
Delete

SolidCAM checks all the geometry entities in the following method:

- SolidCAM compares the SolidWorks model elements geometrically against the relevant entity in the SolidCAM geometry file.

- If SolidCAM determines a misfit between the SolidWorks model and the SolidCAM geometry, this geometry will be marked with a synchronization mark 🚦.

- If there is a gap or overlapping entities, SolidCAM will mark this geometry with a synchronization mark 🚦.

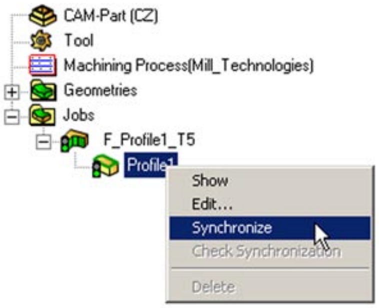

CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
F_Profile1_T5
Profile1

The synchronization mark enables us to see the problematic geometries and solve the not synchronized geometries.

Geometry synchronization

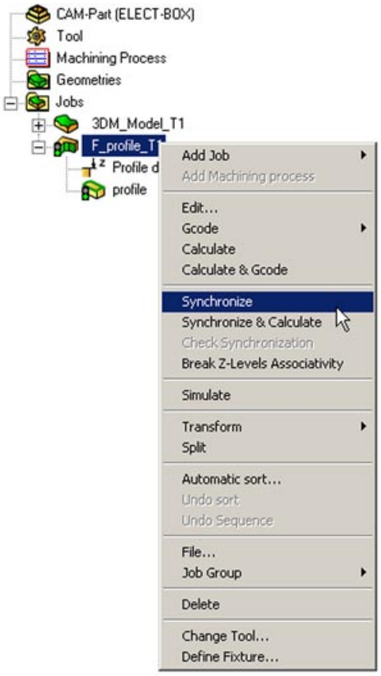
When the geometry has to be synchronized (the geometry marked with the synchronization mark) SolidCAM enables you to do the following steps to synchronize the geometry.

1. Right click on the Geometry name in the SolidCAM Manager.
2. Choose **Synchronize** from the menu.


CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
F_Profile1_T5
Profile1
Show
Edit...
Synchronize
Check Synchronization
Delete

or

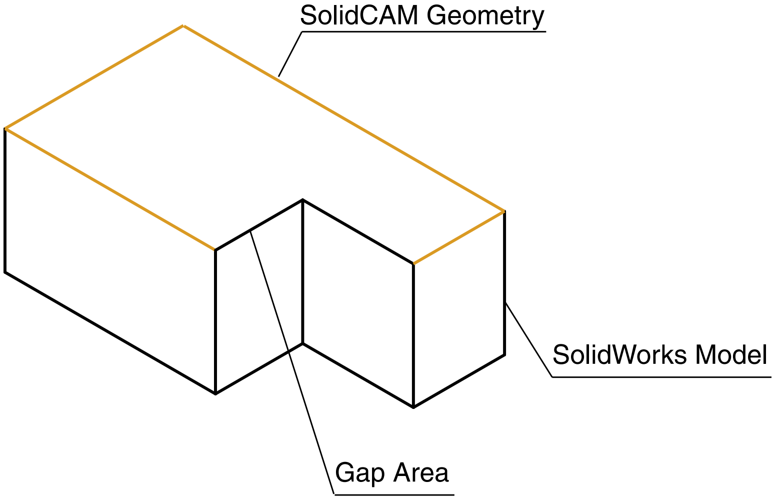
1. Right click on the Job name in the SolidCAM Manager.
2. Choose **Synchronize** (the **Synchronize & Calculate** command enables you to synchronize the Job geometries and calculate the tool path) from the menu.


CAM-Part (ELECT-BOX)
Tool
Machining Process
Geometries
Jobs
3DM_Model_T1
F_profile_T1
Profile d
profile
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...

All the SolidCAM geometries used in the Job will be synchronized.

During the synchronization, SolidCAM tries to update the SolidCAM geometry according to the updated SolidWorks model.

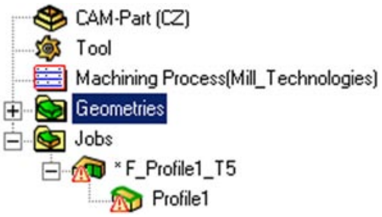
SolidCAM finds the updated edges according to the saved tags data and tries to re-create the geometry chain. SolidCAM recognizes gaps between geometry entities and closes them.



SolidCAM will close the gaps by using the smallest number of entities. SolidCAM has two methods of closing gaps:

- If the original geometry entities are in a plane and the plane is the same as the home plane, SolidCAM will try to close the gap with entities in the same plane only.
- If the original geometry entities are not on a plane, SolidCAM will use a random search algorithm that will close the gap in the geometry. This search algorithm can find different solutions to close the same gap; this depends on the sequence of the entities involved in the loops that connect to the geometry entity.

The geometry that cannot be updated will be marked with an exclamation mark ⚠.



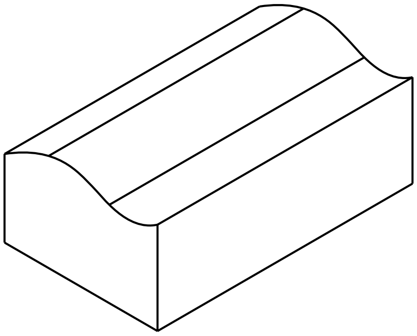
CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
*F_Profile1_T5
Profile1

You have to edit such geometries and manually update the problematic chains.

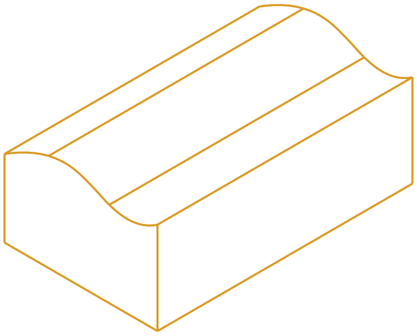
SolidCAM will not find new chains that are not directly connected to old chains. The number of chains in the geometry will stay the same. For example, SolidCAM will not find a new island in a pocket geometry.

4.8.2 Synchronizaton of the 3D Model geometry

The calculation of the tool path for the 3D Model geometry is based on a facet model produced for the whole SolidWorks model.

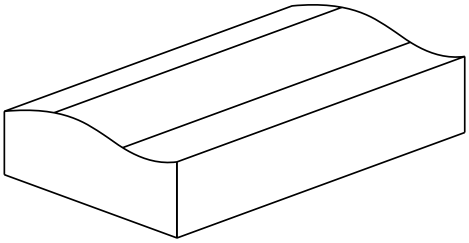


SolidWorks Model

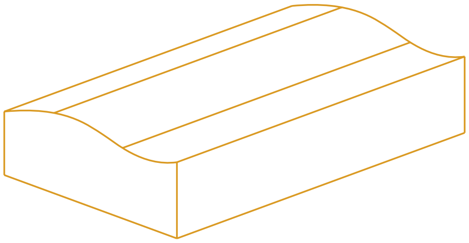


SolidCAM Geometry

When SolidCAM finds that the solid has changed, it does not need to know what has changed; it simply recalculates the facet model.



Updated SolidWorks Model



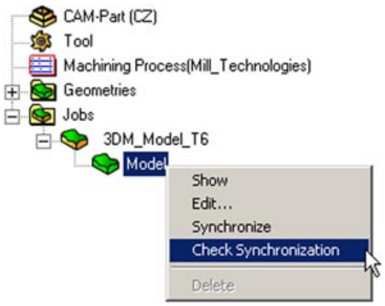
Updated SolidCAM Geometry

Synchronization check

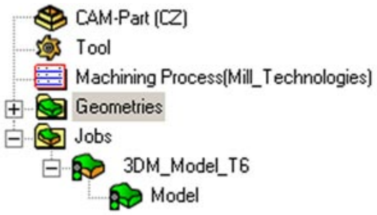
SolidCAM enables you to check the synchronization status by comparing the SolidWorks model and the SolidCAM 3D Model geometry. SolidCAM enables you to check the synchronization status automatically or manually depends on **Synchronization settings**.

When you need to check the synchronization status of the geometry manually, do the following steps:

1. Right click on the Job or Geometry name in the SolidCAM Manager.
2. Choose the **Check Synchronization** item from the menu.


CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
3DM_Model_T6
Model
Show
Edit...
Synchronize
Check Synchronization
Delete

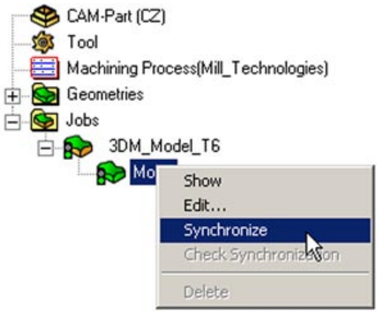
If a difference between them was found, SolidCAM marks the Geometry and related Jobs with the synchronization mark 🚦.


CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
3DM_Model_T6
Model

Geometry synchronization

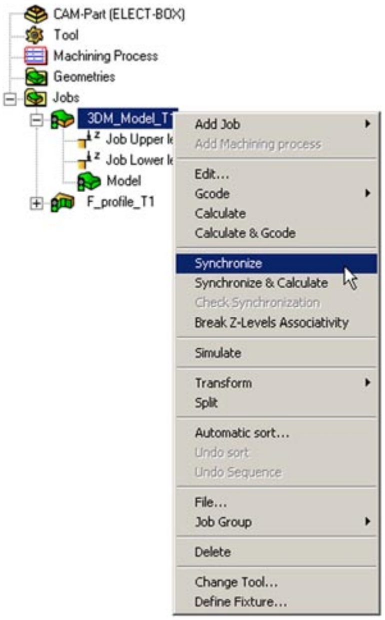
When the geometry have to be synchronized (the geometry marked with the synchronization mark) SolidCAM enables you to do the following steps to synchronize the geometry.

1. Right click on the Geometry name in the SolidCAM Manager.
2. Choose **Synchronize** from the menu.


CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
3DM_Model_T6
Mo
Show
Edit...
Synchronize
Check Synchronization
Delete

or

1. Right click on the Job name in the SolidCAM Manager.
2. Choose **Synchronize** (the **Synchronize & Calculate** command enables you to synchronize the Job geometries and calculate the tool path) from the menu.


CAM-Part (ELECT-BOX)
Tool
Machining Process
Geometries
Jobs
3DM_Model_T
Job Upper l
Job Lower l
Model
F_profile_T1
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...

All the SolidCAM geometries used in the Job will be synchronized.

During the synchronization,SolidCAM recalculates the facet model.

Jobs

5

In SolidCAM, a Job is a single machining operation. A workpiece is usually manufactured using several machining steps and technologies. For each of these steps you can define a separate job.

A job can be very complex, but always uses one tool, one major geometry and executes one machining type, e.g. 3D milling, profile milling or drilling. You can edit any single machining operation, change the job sequence and generate G-Code, combining and splitting the job-list of your CAM-Part.

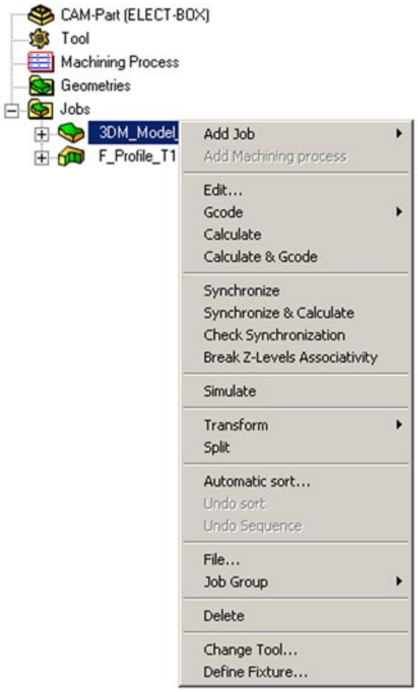
❐ Adding a Job
❐ Job interface
❐ Working with Jobs
❐ Job Transformations
❐ Fixture

5.1 Adding a Job

This command enables you to add a new job to your CAM-Part. In SolidCAM a Job is a single machining operation. Normally you manufacture a workpiece using several machining steps and technologies. For each of these steps you can define a separate job. A job can be very complex, but always uses one tool, one major geometry and executes one machining type, e.g. 3D milling, profile milling or drilling.

You can edit any single machining operation, change the job sequence and generate G-Code, combining and splitting the job-list of your CAM-Part.

1. In the CAM Manager, move the mouse on the Job field and press the **right** mouse button. The **Job menu** is displayed.

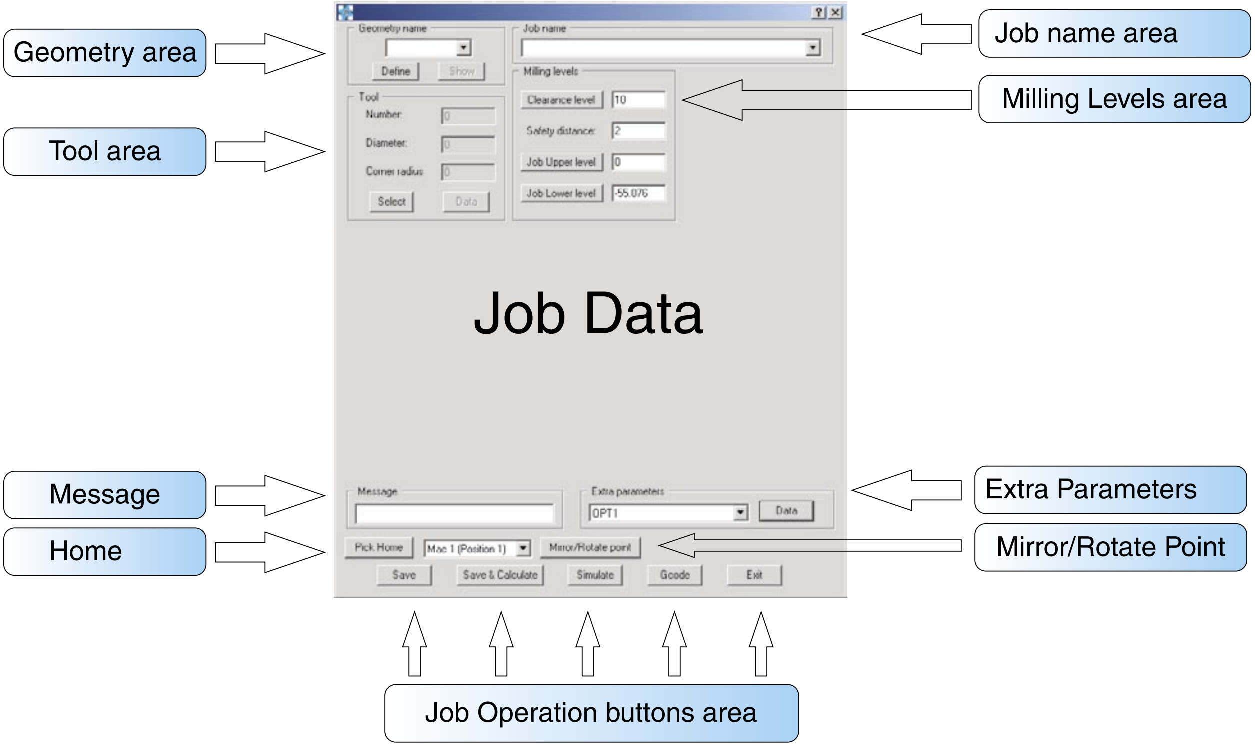

CAM-Part (ELECT-BOX)
Tool
Machining Process
Geometries
Jobs
3DM_Model
F_Profile_T1
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...

2. Move to the **Add** sub-menu and select the type of job you want to define.

- When you add a Job after right clicking on the Job symbol, it will be added at the end of the job list.
- If you execute the Add Job command on an existing job, the new job will be inserted after the selected job.

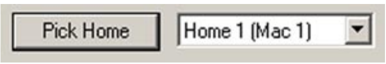
5.2 SolidCAM Job Interface

Every SolidCAM Job has common areas. Click on the area name on the picture below for a detailed explanation.

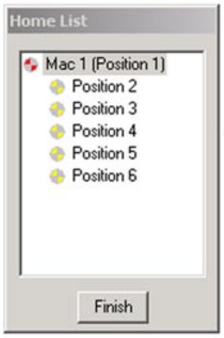

Geometry area
Tool area
Message
Home
Job name area
Milling Levels area
Extra Parameters
Mirror/Rotate Point
Job Operation buttons area
Job Data
Geometry name
Define
Show
Tool
Number
Diameter
Corner radius
Select
Data
Job name
Milling levels
Message
Extra parameters
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

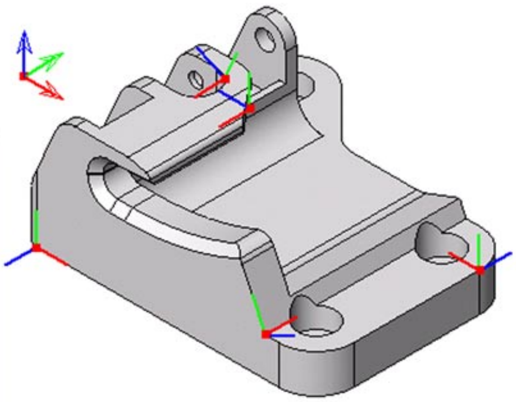
Clearance level: 10
Safety distance: 2
Job Upper level: 0
Job Lower level: -55.076
Extra parameters: OPT1
Home: Mac 1 (Position 1)

5.2.1 Home

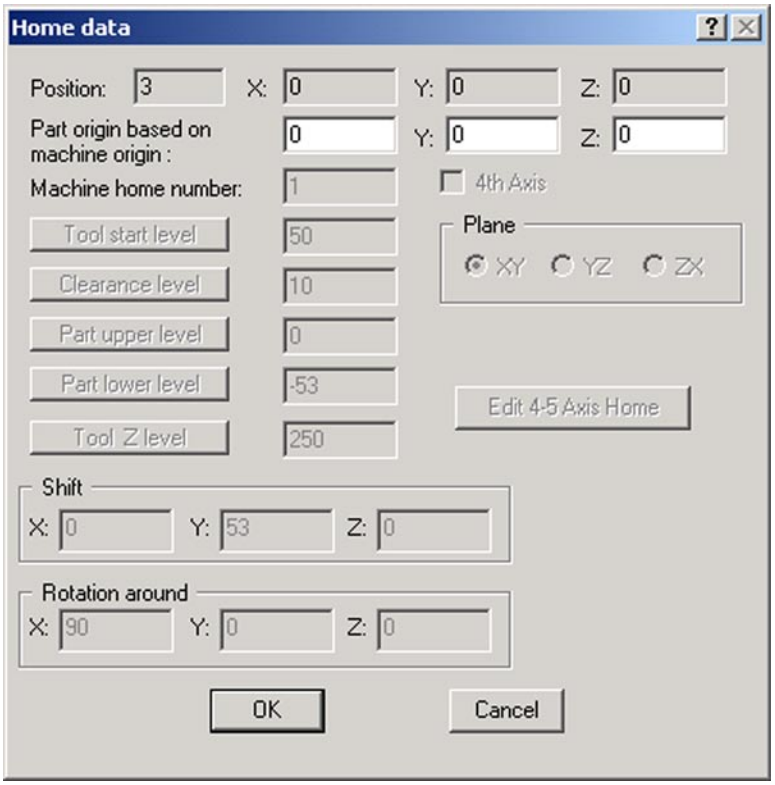

Pick Home
Home 1 (Mac 1)

Define the home number. Type the home number or select it from the graphic screen by pressing the **Home number** button. The **Home List dialog** will be displayed.


Home List
Mac 1 (Position 1)
Position 2
Position 3
Position 4
Position 5
Position 6
Finish



To get more information about the Home right click on the Home name and choose the **Inquire** option from the menu. The **Home data** will be displayed.


Home data
?
Edit 4-5 Axis Home
OK
Cancel

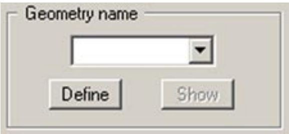
Position: 3 X: 0 Y: 0 Z: 0
Part origin based on machine origin : 0 Y: 0 Z: 0
Machine home number: 1
☐ 4th Axis
Tool start level 50
Clearance level 10
Part upper level 0
Part lower level -53
Tool Z level 250
Plane ● XY ○ YZ ○ ZX
Shift X: 0 Y: 53 Z: 0
Rotation around X: 90 Y: 0 Z: 0

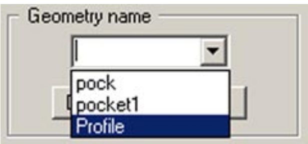
After the Home selection, the model will by rotated to the selected home orientation.

The **Home selection** operation must be the first step in the Job definition process.

5.2.2 Geometry

In this section, specify the main geometry for the Job.


Geometry name
Define
Show


Geometry name
pock
pocket1
Profile

If you have already defined geometries for this CAM-Part, you can select a geometry from the **Geometry name** list field.

- **Show** will display the model geometry in the Host CAD.
- **Define** enables you to define a new geometry for the Job.

5.2.3 Tool

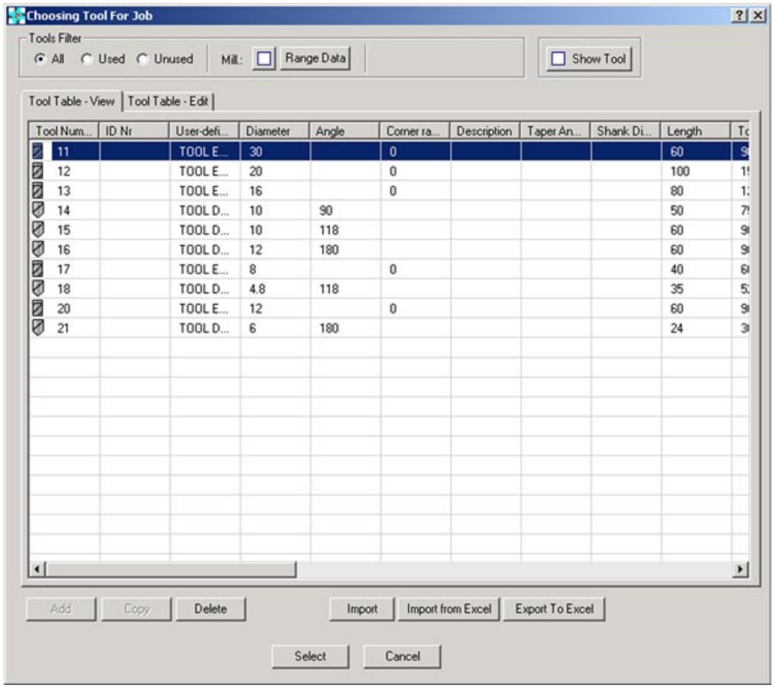
In the tool area of the Job screen, three major tool parameters are displayed:

- **Number**
- **Diameter**
- **Corner radius**

Select

This button enables you to edit tool parameters or define the tool you want to use for this job.

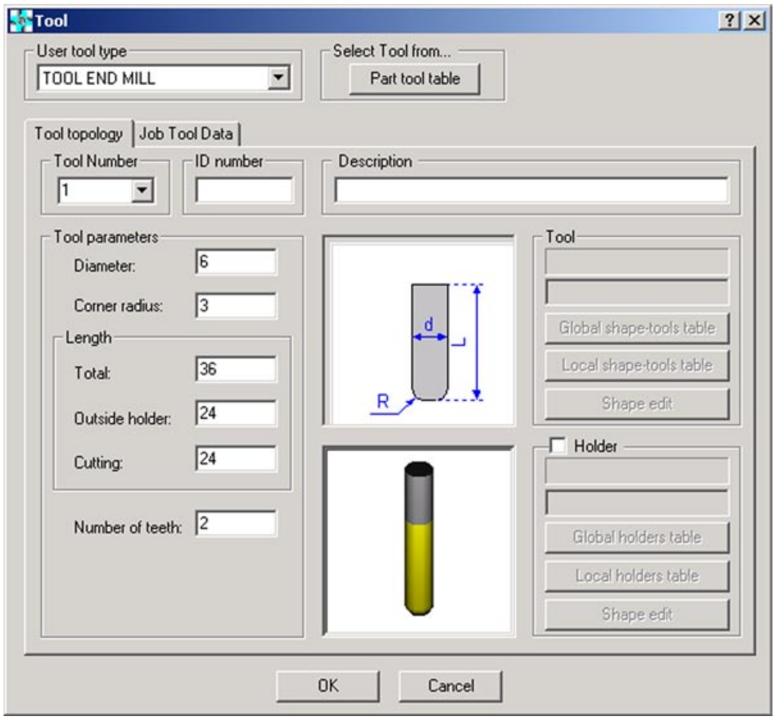
- When the tool for the job is not defined, this button displays the **Part Tool Table dialog** which enables you to choose the tool from the Part Tool table.


Choosing Tool For Job
Tools Filter
● All
○ Used
○ Unused
Mill: ☐
Range Data
☐ Show Tool
Tool Table - View
Tool Table - Edit
Add
Copy
Delete
Import
Import from Excel
Export To Excel
Select
Cancel

Tool Num...	ID Nr	User-defi...	Diameter	Angle	Corner ra...	Description	Taper An...	Shank Di...	Length	Tc
11		TOOL E...	30		0				60	9
12		TOOL E...	20		0				100	1
13		TOOL E...	16		0				80	1
14		TOOL D...	10	90					50	7
15		TOOL D...	10	118					60	9
16		TOOL D...	12	180					60	9
17		TOOL E...	8		0				40	6
18		TOOL D...	4.8	118					35	5
20		TOOL E...	12		0				60	9
21		TOOL D...	6	180					24	3

Choose the necessary tool from the Part Tool Table and click on the **Select** button. The tool will be chosen for the Job. The **Tool dialog** will be displayed with the parameters of the chosen tool.

- When the tool for the job is defined, this button displays the **Tool dialog**. This dialog enables you to edit the tool parameters. You can also choose tools from the Part Tool table.


Tool
Select Tool from...
Part tool table
Tool topology
Job Tool Data
ID number
Description
d
L
R
Tool
Global shape-tools table
Local shape-tools table
Shape edit
Global holders table
Local holders table
Shape edit
OK
Cancel

User tool type: TOOL END MILL

Tool Number: 1

Tool parameters

Diameter: 6

Corner radius: 3

Length

Total: 36

Outside holder: 24

Cutting: 24

Number of teeth: 2

☐ Holder

This dialog enables you to edit the tool definition.

User Tool Type

This field shows the tool type.

The tool types were defined with the **Tool type names dialog**.

These types are the internal tool types. SolidCAM checks the possibility of a specific internal tool type to work in the selected job.

Selecting a shaped tool will activate the **Tool name** section on the **Tool topology** page.

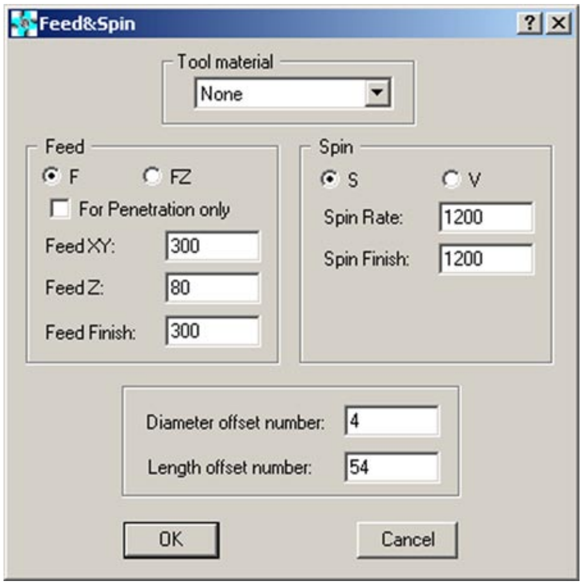
Part Tool table

This button enables you to choose the tool from the Part Tool table. The **Part Tool Table dialog** will be displayed.

- The **Topology page** (see the topic **3.8.2.**) contains the Topology Data of the tool like diameter, length, angle, corner radius etc.
- The **Job Data page** (see the topic **3.8.3.**) contains the default Tool Material data, Feed and Spin data and diameter and length offset numbers.

Data

The **Data** button displays the **Feed&Spin dialog**. This dialog displays the tool material, feed and speed parameters and the diameter and length offset numbers used in the current job. When the tool is selected for the job, SolidCAM fills this dialog with the default data of the selected tool. The **Tool Data dialog** enables you to edit the tools for the specific job only.



Tool material: None

Feed: ● F ○ FZ

☐ For Penetration only

Feed XY: 300

Feed Z: 80

Feed Finish: 300

Spin: ● S ○ V

Spin Rate: 1200

Spin Finish: 1200

Diameter offset number: 4

Length offset number: 54

Tool Material

The **Tool Material** is used to describe one or more of the following properties of the tool:

- Material of the tool (High Speed Steel, Carbide etc…)
- Working conditions of the tool
- Use of tool (rough, finish, etc…)

The **Tool Material** is used to define the **Speed and Feed defaults**.

When you choose the **Tool material**, the feed type will be changed to **Fz** and the spin type to **V**. The **Feed** and **Spin** values are default system values since the **Work material** has not been defined yet. Loading the tool to a job will connect the **Tool material** to the **Work material** and the correct **Feed** and **Spin** values will be loaded from the **Speed/Feed defaults table**.

Spin

This field defines the spinning speed of the tool. It defines two spin values:

- **Spin Rate** - Normal spin-rate; used in rough milling.
- **Spin Finish** - Finish spin-rate; used in finish milling.

The spin value can be defined in two types of units, namely **S** and **V**.

S is the default and it signifies Revolutions per Minute. **V** signifies Material cutting speed in Meters/Minute in the Metric system or in Feet/Minute in the Inch system; it is calculated according to the following formula:

V = (S * PI * Tool Diameter) / 1000

Feed

This field defines the feed rate of the tool. It defines three feed values:

- **Feed XY** - Feed-rate in the XY plane.
- **Feed Z** - Feed-rate in the Z direction.
- **Feed Finish** - Feed-rate used for finish milling.

The feed value can be defined in two types of units, namely: **F** and **Fz**.

F is the default and it signifies Units per minute. **Fz** signifies Units per tooth and is calculated according to the following formula:

Fz = F/(Number of Teeth * S)

For Penetration only

When this option is turned on, the defined **Feed Z** will be used for vertical movements only.

When this option is turned off, the defined **Feed Z** will be used for all movements when the Z-coordinate changes.

This option is active only for the **3D Model Job**.

Diameter Offset Number

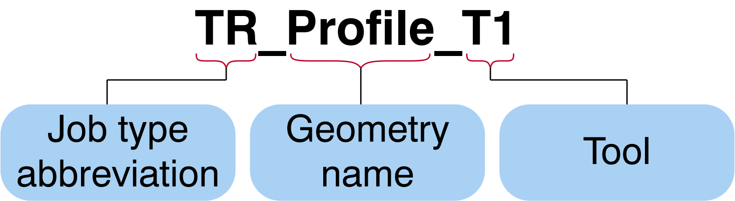
This parameter defines the number of the Diameter Offset Register of the current tool in the Offset table of the CNC machine.

Length Offset number

This parameter defines the number of the Length Offset Register of the current tool in the Offset table of the CNC machine.

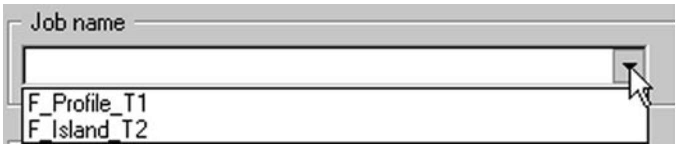
5.2.4 Job name

Once you select a geometry and a tool, SolidCAM will automatically enter a default name (e.g. 3DM_Model_T2) based on the job type, geometry name and tool number. You can then change the job name, if necessary.

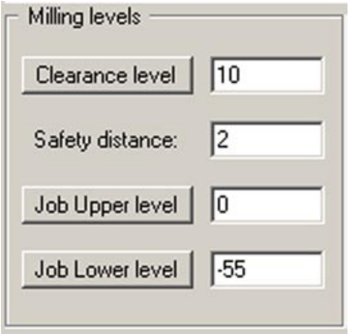

TR_Profile_T1
Job type abbreviation
Geometry name
Tool

When you add a new job, you can use an existing job as a template for the new job. All fields of the template job will be copied and you only need to edit some changing parameters, e.g. select another geometry or surface offset.

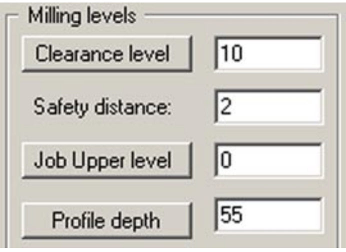
Use the **Job name** list field to select the template job.


Job name
F_Profile_T1
F_Island_T2

5.2.5 Milling Levels



Clearance level: 10
Safety distance: 2
Job Upper level: 0
Job Lower level: -55



Clearance level: 10
Safety distance: 2
Job Upper level: 0
Profile depth: 55

In this section you must specify the Z-levels in which the tool movements and the job will be executed. The default input to this fields are the CAM-Part values you have specified in the **Home data dialog**.

Enter the values or use the respective buttons to pick the Z-levels from your 3D model.

- **Clearance level** is the Z-level to which the tool will retreat when moving from cut to cut.
- **Safety distance** is the distance to the Job upper plane at which the tool will start moving at the Z feed-rate you have entered for the tool. Movements from the Job Clearance plane to this height are performed in rapid move.
- **Job Upper level** is the Z-level at which the machining will start in cutting operations.
- **Job Lower level** or **Depth** sets the Z-level below which the tool will not mill. SolidCAM will not penetrate this plane in any milling strategy.

The value of these parameters is the Z-coordinate. These parameters can be defined either by entering the value in the edit box or by picking a specific point on the solid model (except Safety distance). In the latter case, the parameters are associative to the solid model. Associativity enables SolidCAM to be synchronized with the solid model changes; SolidCAM automatically updates the CAM data when the model is modified.

The following Jobs parameters are associative to the SolidWorks model.

- **Clearance level**
- **Job Upper Level**
- **Job Lower Level**

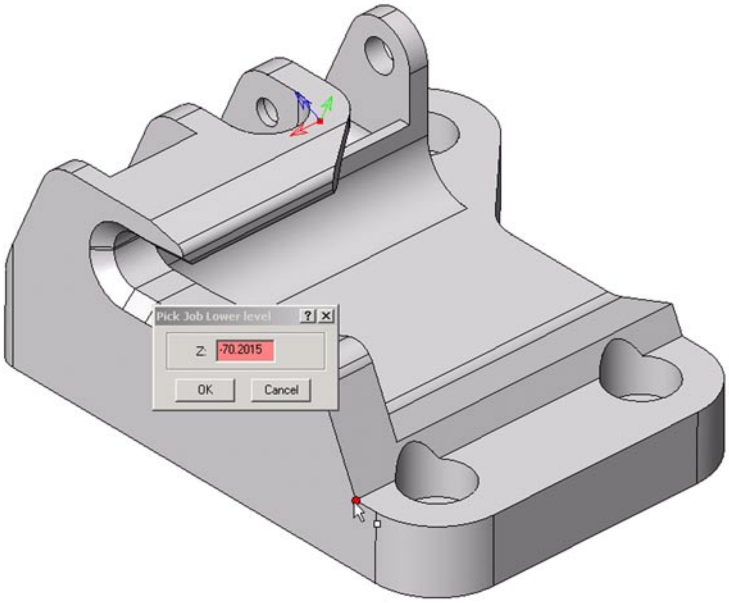
SolidCAM enables you to define these parameters by picking the solid model.

The **Depth** parameter used in some Jobs (Profile Job, Pocket Job etc) is indirectly associative. You have two possibilities:

- To define the **Depth** by manual input.
- To define the **Job Lower Level** by picking on the model. In this case the **Depth** will be calculated automatically. In this case the associativity will be established in the **Job Lower Level**. When either the **Job Upper Level** or the **Job Lower Level** is synchronized, the **Depth** will be updated.

The associativity will be saved only for the following picked items:

- Vertex
- Sketch Point
- Reference point
- Part Origin
- Planar Face parallel to the XY plane of the current Home
- Plane parallel to the XY plane of the current Home



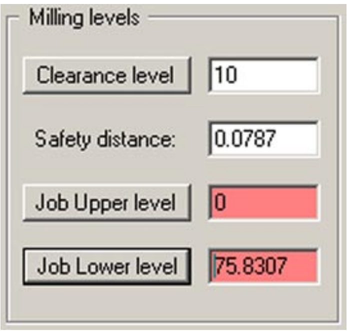
Pick Job Lower level

Z: -70.2015

The **Pick dialog** displays the Z-coordinate of the picked point.

If the value is associative to the model, the edit box will be highlighted in orange.

When the parameter is defined by on-model pick, SolidCAM displays the Z-coordinate of the picked point in the related edit box. The background of the edit box will be changed. The value displayed in this edit box is associative to the solid model.


Milling levels

Clearance level: 10

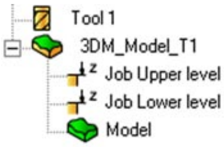
Safety distance: 0.0787

Job Upper level: 0

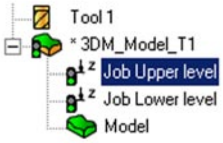
Job Lower level: 75.8307

If you type in this edit box, the associativity is removed.

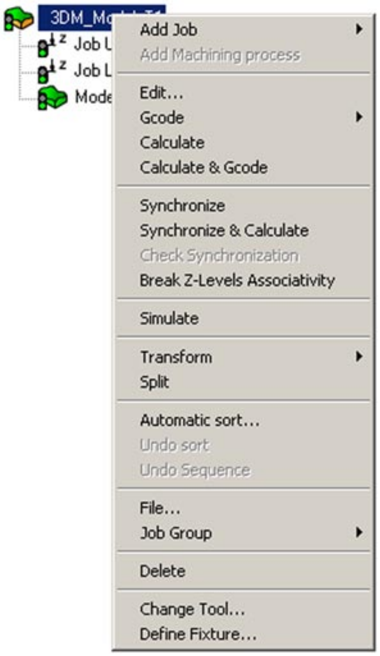
The associative parameters are displayed in the CAM Manager under the Job name.

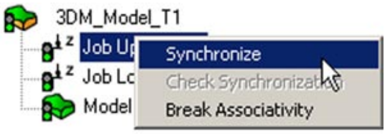

Tool 1
3DM_Model_T1
Job Upper level
Job Lower level
Model

When the parameters are not synchronized, the parameters and the parent Job are marked with the synchronization mark.

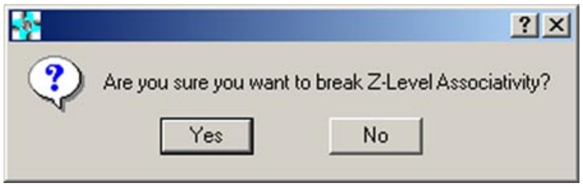

Tool 1
* 3DM_Model_T1
Job Upper level
Job Lower level
Model

The synchronization commands are available for jobs in the right-click menu.


3DM_Model_T1
Job L
Job L
Mode
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...

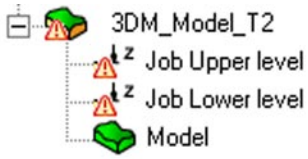

3DM_Model_T1
Job Up
Job Lo
Model
Synchronize
Check Synchronization
Break Associativity

- The **Synchronize** button enables you to synchronize the value of the parameters with the updated model.
- The **Synchronize & Calculate** command enables you to synchronize the parameters value with the updated model and calculate job after the synchronization.
- The **Check Synchronization** button enables you to check the model for updates affecting the specified parameter.
- The **Break Z-Levels Associativity** / **Break associativity** command enables you to break the associativity between Job Levels parameters and the solid model. The confirmation message will be displayed.



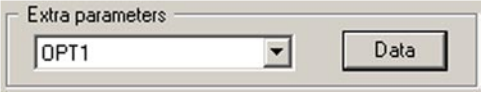
Are you sure you want to break Z-Level Associativity?

When the synchronization fails, SolidCAM marks the job and the parameters with an exclamation mark.


3DM_Model_T2
Job Upper level
Job Lower level
Model

The parameters definition can be edited within the Job dialog. You have to either type the necessary values or redefine the points on the model. In the first case, the associativity will be removed.

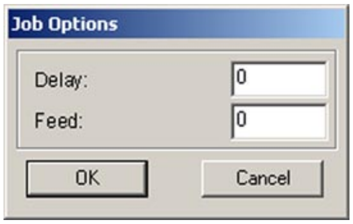
5.2.6 Extra parameters


Extra parameters
Data

OPT1

This field will only be activated when special job options have been implemented in the post-processor you use for this CAM-Part.

Click on the **Data** button. The **Job option dialog** will be shown with the additional parameters defined in the post-processor.

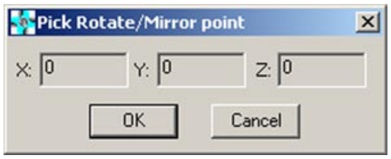

Job Options
OK
Cancel

Delay: 0

Feed: 0

5.2.7 Mirror/Rotate point

If you use a Job transformation, this button enables you to define the point around which this job will be rotated or mirrored. The **Mirror/Rotate point dialog** will be displayed, showing you the coordinates of the current mirror/rotate position.


Pick Rotate/Mirror point
OK
Cancel

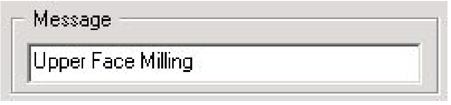
X: 0 Y: 0 Z: 0

1. Enter the X,Y,Z coordinates into the Edit bar and confirm with Enter or pick a position using the CAD point selection options.

2. Confirm with **OK**.

5.2.8 Message

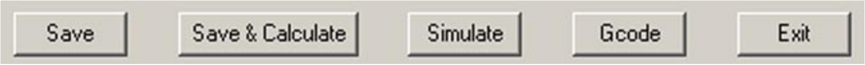
You can type a message into this field that will appear in the generated G-Code file.



Upper Face Milling

```
G43G0 X-49.464 Y-38.768 Z12. S1000 M3
(Upper Face Milling)
(-------------------------)
(P-POCK-T2 - POCKET)
(-------------------------)
G0 X-49.464 Y-38.768
Z10.
```

5.2.9 Job operation buttons



- **Save**

All settings and parameters of the job will be saved in your disk. The tool path, however, will not be calculated. This enables you to postpone the calculation or to calculate jobs in batches.

Use the corresponding commands of the CAM Manager to calculate the job later on.

In the job list, all jobs that have not been calculated are marked with an asterix (*).

- **Save&Calculate**

All settings and parameters of the job will be saved to your disk and the tool path will be calculated. After the calculation has been performed, you can exit the Job screen and return to the CAM Manager.

- **Simulate**

Performs **simulation** of the cutting process.

The Job must be calculated before this option can be used.

- **G-Code**

  Generates **G-Code** for the Job.

  The Job must be calculated before this option can be used.

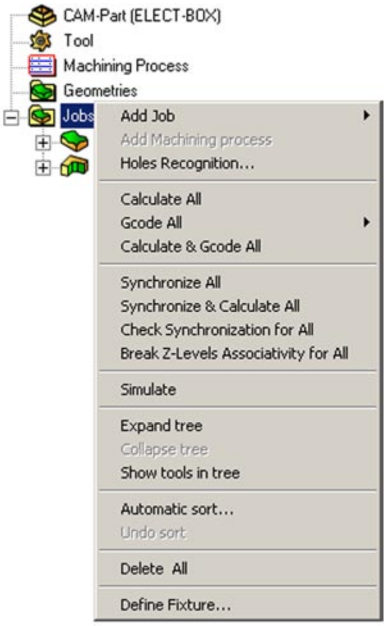
- **Exit**

  This field enables you to exit the job screen and return to the CAM Manager. Any changes you made in the job screen will be lost if you exit the job without prior saving.

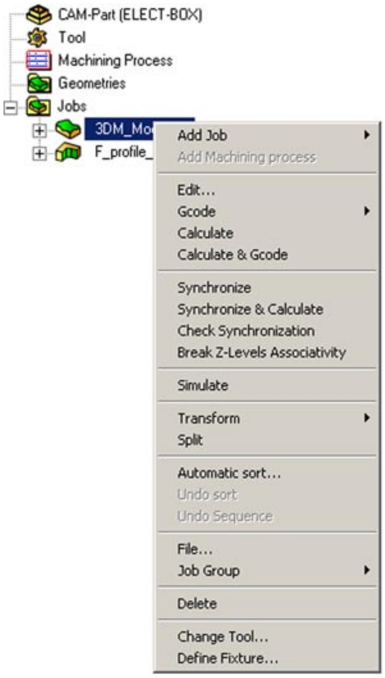
5.3 Working with Jobs

Some of the Job commands will affect all existing jobs; others can only be applied to one or more selected jobs. SolidCAM will produce different menus, depending on whether you **right** mouse click on the Job symbol of the CAM Manager or on an existing job in the job list of your CAM-Part.

In the CAM Manager move the mouse **on the Job symbol** and press the **right** mouse button. The **Job menu** will be displayed.


CAM-Part (ELECT-BOX)
Tool
Machining Process
Geometries
Jobs
Add Job
Add Machining process
Holes Recognition...
Calculate All
Gcode All
Calculate & Gcode All
Synchronize All
Synchronize & Calculate All
Check Synchronization for All
Break Z-Levels Associativity for All
Simulate
Expand tree
Collapse tree
Show tools in tree
Automatic sort...
Undo sort
Delete All
Define Fixture...

If you press the **right** mouse button **on an existing job**, the **Job commands** are displayed.

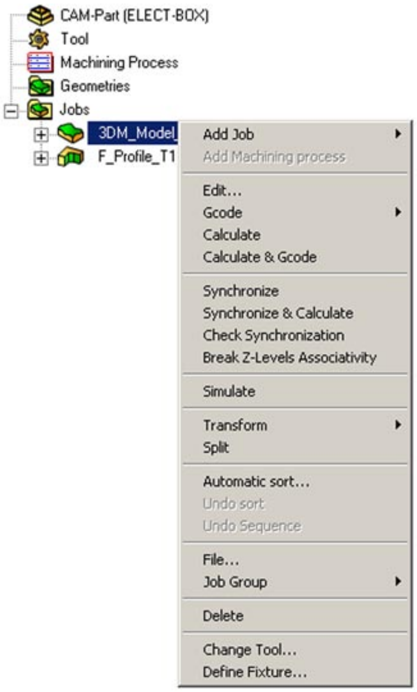

CAM-Part (ELECT-BOX)
Tool
Machining Process
Geometries
Jobs
3DM_Mo
F_profile_
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...

5.3.1 Add Job

This command enables you to add a new job to your CAM-Part. In SolidCAM a Job is a single machining operation. Normally you manufacture a workpiece using several machining steps and technologies. For each of these steps you can define a separate job. A job can be very complex, but always uses one tool, one major geometry and executes one machining type, e.g. 3D milling, profile milling or drilling.

You can edit any single machining operation, change the job sequence and generate G-Code, combining and splitting the job-list of your CAM-Part.

1. In the CAM Manager, move the mouse on the Job field and press the **right** mouse button. The **Job menu** is displayed.


CAM-Part (ELECT-BOX)
Tool
Machining Process
Geometries
Jobs
3DM_Model
F_Profile_T1
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...

2. Move to the **Add** sub-menu and select the type of job you want to define.

- When you add a Job after right clicking on the Job symbol, it will be added at the end of the job list.
- If you execute the Add Job command on an existing job, the new job will be inserted after the selected job.

5.3.2 Add Machining process

This command enables you to use a previously created **Machining process** in your CAM-Part.

For more information on the **Machining Processes**, please refer to the topic **11.**

5.3.3 Edit

This command enables you to view or change the definition and parameters of an existing job. If you select this command, the corresponding job screen will be displayed and you can change the job definition.

5.3.4 Calculate/Calculate All

Before you will be able to generate G-Code for your CNC-machine, you have to perform the tool path calculation. Based on the geometries, machining strategy and technological parameters you have defined and saved in the Job, SolidCAM will calculate the movements of the tool.

If you expect a long tool path calculation time, mainly in 3D milling jobs, it is advisable to define your jobs, save them and postpone the tool path calculation. This function enables you to calculate large CAM-Parts with many jobs in a batch routine e.g. overnight.

5.3.5 G-Code/G-Code All

The commands in this sub-menu enable you to generate, list, copy or print NC-programs.

For more information on the **Generating G-code**, please refer to the topic **9**.

5.3.6 Calculate & G-Code All

This command enables you to calculate all the Jobs in your CAM-Part and produce a G-Code.

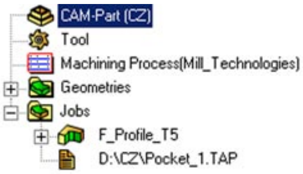
5.3.7 Simulate

You can simulate and verify jobs that have been saved and calculated.

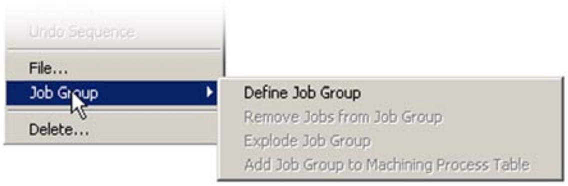
For more information on the **Simulation**, please refer to the topic **8**.

5.3.8 File

This option enables you to append some ASCII files with a G-Code to your CAM-Part.

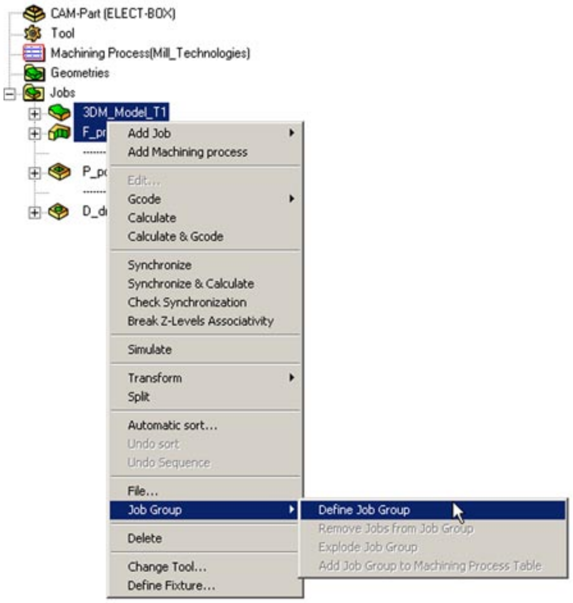

CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
F_Profile_T5
D:\CZ\Pocket_1.TAP

5.3.9 Job Group

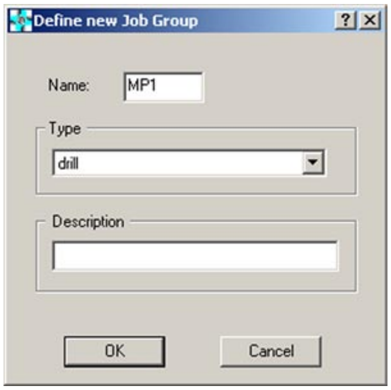

Undo Sequence
File...
Job Group
Delete...
Define Job Group
Remove Jobs from Job Group
Explode Job Group
Add Job Group to Machining Process Table

The Job Group in SolidCAM is the set of separate jobs with a defined order in the sequence. The Job Group menu enables you to control Job Groups.

Define Job Group


CAM-Part (ELECT-BOX)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
3DM_Model_T1
F_pr
P_pc
D_dr
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...
Define Job Group
Remove Jobs from Job Group
Explode Job Group
Add Job Group to Machining Process Table

These options convert the selected Jobs into a Job Group. The **Define new Job Group dialog** will be displayed.


Define new Job Group
? X
OK
Cancel

Name: MP1

Type: drill

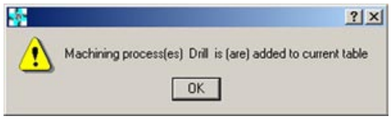
Description:

1. Enter the **Job Group name** in the name field.
2. Choose the **type** from the list.
3. You can also enter the **Description** - a text string that describes the Job Group.

Selected Jobs will be converted into the Job Group.

Add Job Group to Machining Process table

This option enables you to insert the selected **Job Group** into the current **Machining process table**.



Machining process(es) Drill is (are) added to current table

Remove Jobs from Job Group

This option removes the selected Job from the Job Group.

Explode Job Group

This option explodes the Job Group into single Jobs.

5.3.10 Delete/Delete all

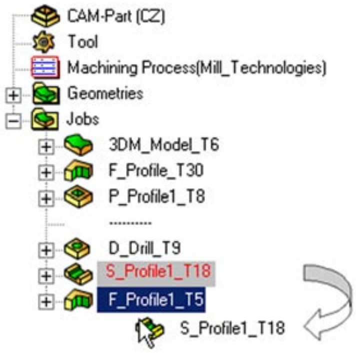
This command enables you to delete jobs from the job list of your CAM-Part.

1. Select a job using the mouse. To delete several jobs at a time, select them using the Shift and Ctrl-keys.
2. With the right mouse button, click on the jobs and select the Delete command.
3. To delete several jobs at a time, select „Yes to all“ in the confirmation message.

5.4 Managing Jobs in the CAM-Manager tree

5.4.1 Job Sequence

You can change and re-arrange the sequence of the jobs in the CAM Manager. Simply drag the job to its new place in the **job list**.

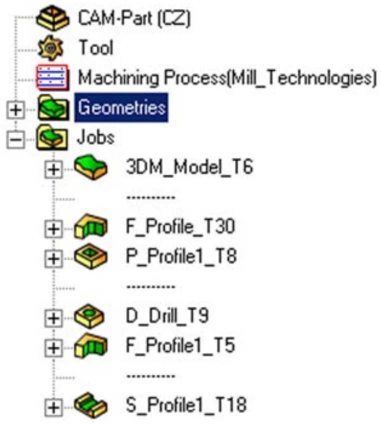

CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
3DM_Model_T6
F_Profile_T30
P_Profile1_T8
....
D_Drill_T9
S_Profile1_T18
F_Profile1_T5
S_Profile1_T18

5.4.2 Undo Sequence

This command enables you to undo your last change in the sequence of the job list.

5.4.3 Split

This command enables you to separate your list of jobs before the G-Code generation. If you place split marks after jobs and select G-Code All, separate NC-programs will be generated for each job or group of jobs that are located between the corresponding split mark. In this **example**, you will receive 5 separate G-Code files for the jobs divided by the split marks.


CAM-Part (CZ)
Tool
Machining Process(Mill_Technologies)
Geometries
Jobs
3DM_Model_T6
....
F_Profile_T30
P_Profile1_T8
....
D_Drill_T9
F_Profile1_T5
....
S_Profile1_T18

Adding Split marks

1. Click on the job after which you want to insert a split mark.
2. Press the right mouse button and select the **Split** command.
3. The split mark will be inserted.

Removing Split marks

1. Click on a split mark and press the right mouse button.
2. Select the **Delete** command and confirm the message with **Yes**.

Just like re-sequencing jobs, you can move a split mark to another location in the job list. Simply drag the split mark on the job after which you want to insert it.

5.4.4 Expand

This command enables you to open the job list, so that all the geometries used by each job will be displayed in the tree structure of the CAM Manager.

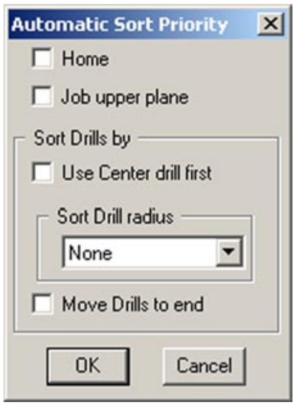
5.4.5 Collapse

This command enables you to reduce the job list, so that only the job names will be displayed in the tree structure of the CAM Manager.

5.4.6 Automatic Sort

Automatic sort enables you to sort all of the jobs that you created to work in a specific order.

1. Choose the **Automatic sort** item from the menu. The **Automatic Sort Priority** dialog is displayed.



Automatic Sort Priority

☐ Home

☐ Job upper plane

Sort Drills by

☐ Use Center drill first

Sort Drill radius: None

☐ Move Drills to end

2. Choose the desired options.

3. Click on the **OK** button to confirm the operation.

4. All the Jobs will be sorted in the specified order.

If you press **OK** without choosing any of the options in the dialog window, the jobs will be sorted starting from tool number 1 and upward.

Automatic Sort options:

- **Home**: Jobs will be sorted according to the home number.

- **Job upper plane**: Jobs started on a higher plane will come before jobs started on a lower plane.

- **Use center drill first**: All tools that were tagged as center drills will be moved to the top of the job list.

- **Sort drill radius**: Drills can be sorted according to size; either from small diameters to large diameters or from large diameters to small diameters.

- **Move drills to end**: This option will move all drills that are not tagged as center drills to the end of the job list.

The following is the order of precedence if more than one option is chosen.

1. Home
2. Job upper plane
3. Use Center drill first
4. Sort Drill radius
5. Move Drills to end

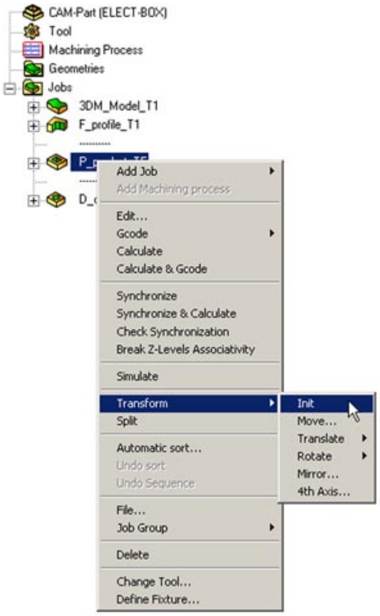
Undo Sort

This option will return the automatic sort to the original order.

5.5 Job Transformations

When you transform a Job or a group of jobs, commands such as move, copy, translate and rotate will insert the corresponding G-function and cycle into the G-Code file. You can check the defined transformation during the tool path simulation.

In the CAM Manager, move the mouse **on a Job** and press the **right** mouse button. Move the mouse to the **Transform sub-menu**.


CAM-Part (ELECT-BOX)
Tool
Machining Process
Geometries
Jobs
3DM_Model_T1
F_profile_T1
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...
Init
Move...
Translate
Rotate
Mirror...
4th Axis...

- You can perform only one transform action on each job.
- The job transformations will only work if your CNC-machine controller supports the corresponding function.

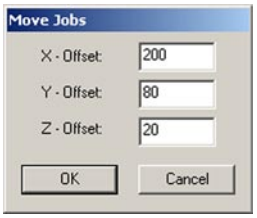
5.5.1 Init

Init will cancel and reset any transformation that was performed on the Job.

5.5.2 Move

This option moves a Job or a group of Jobs while offsetting them in three axes, namely: X, Y and Z.

When you choose this option, the **Move dialog** box is displayed:



X - Offset: 200
Y - Offset: 80
Z - Offset: 20

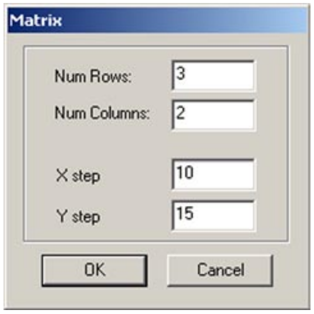
1. Enter the offset values for the X, Y and Z coordinates.
2. Confirm by clicking on **OK**.

5.5.3 Translate

This option copies a Job or a group of Jobs according to a Matrix or List of points.

- **Matrix**

  Copies the Job to a rectangular matrix of locations. When you choose this option, the **Matrix dialog** box is displayed.

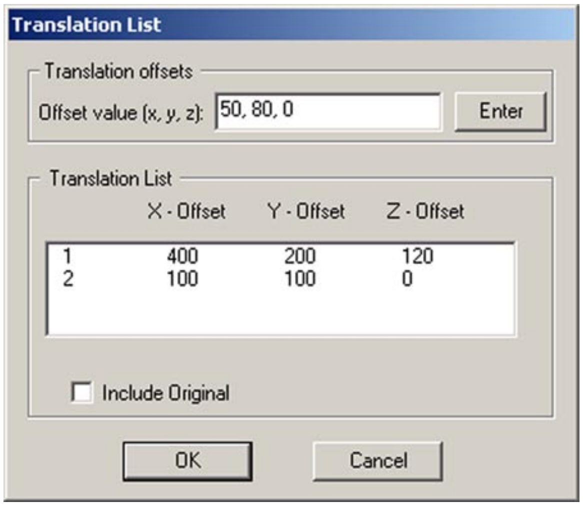


Num Rows: 3
Num Columns: 2
X step 10
Y step 15

1. **Num Rows** - enter the number of Rows.
2. **Num Columns** - enter the number of Columns.
3. **X Step** - enter the delta value between each two columns.
4. **Y Step** - enter the delta value between each two rows.
5. Confirm with **OK**.

- **List**

Copies the Job to a list of Locations defined by their **X**, **Y** and **Z** values. The Home of the Job will be moved to the absolute X,Y,Z positions defined in the **Translation list dialog**.



Offset value (x, y, z): 50, 80, 0

	X - Offset	Y - Offset	Z - Offset
1	400	200	120
2	100	100	0

☐ Include Original

1. Enter the values of the absolute **X**,**Y**,**Z** position where the Job should be executed (Separate values by comma or blank).
2. Click Enter to add the position to the **Translation list**.
3. Check the **Include Original** box, if you want to execute the job in its original position before translating it.
4. Confirm with **OK**.

- To edit a position in the translation list, enter the new values for this position into the Edit field, click on the position you want to overwrite and then click on **Enter**.
- To delete a position from the translation list, click on the position and press the **Del**-key.

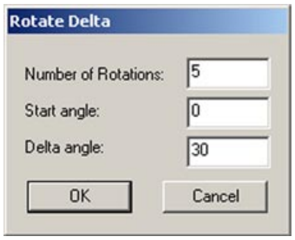
5.5.4 Rotate

This option copies a Job or a group of Jobs while rotating them. Two options are available: **Delta** and **List**.

The point about which the rotation is executed is either the home position or the **Mirror/Rotate point** defined in the job.

- **Delta**

A number of rotations are performed from a start angle with a uniform delta angle. The **Rotate dialog** box is displayed.



Rotate Delta

Number of Rotations: 5

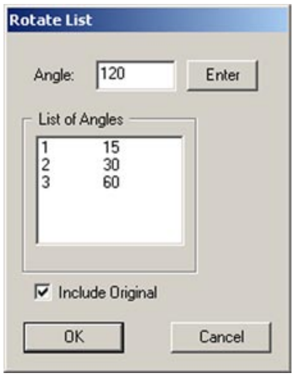
Start angle: 0

Delta angle: 30

1. Enter the **Number of Rotations**.
2. **Start angle** - enter the start angle (+/-) relative to the positive X-axis.
3. **Delta angle** - enter the delta value (+/-) between two rotation steps.
4. Confirm with **OK**.

- **List**

Copies the Job to a list of Angle Locations defined by their **relative Angle to the positive X-axis**. The **Rotations list dialog** will be displayed.



Rotate List

Angle: 120

List of Angles

1	15
2	30
3	60

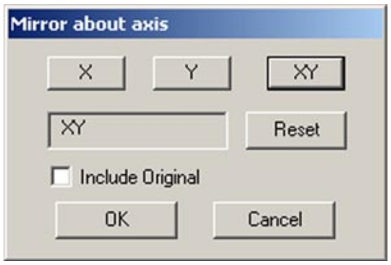
☑ Include Original

1. Enter the value of the angle (+/-) where the Job should be executed.
2. Click on **Enter** to add the angle to the list of angles.
3. Check the **Include Original** box, if you want to execute the job in its original position before rotating it.
4. Confirm with **OK**.

- To edit an angle in the angles list, enter the new value for this angle into the Edit field, click on the angle you want to overwrite and then click on **Enter**.
- To delete an angle from the angle list, click on the angle and press the **Del**-key.

5.5.5 Mirror

This option mirrors a Job or a group of Jobs around the X, Y or both axes. The **Mirror dialog** will be displayed.

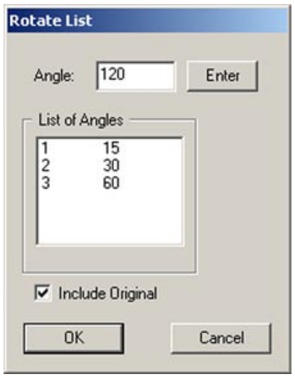


The point around which the mirror operation will be executed is either the home position or the **Mirror/Rotate point** defined in the job.

1. Click on the corresponding button to define the axis around which the jobs will be mirrored:
    - **X** - mirrors around the X -axis.
    - **Y** - mirrors around the Y -axis.
    - **XY**- mirrors around both the X and Y-axes.
    - **Reset** enables you to clear the Entry field.
2. Check the **Include Original** box if you want to execute the job in its original position before mirroring it.
3. Confirm with **OK**.

5.5.6 4th Axis

This option enables you to execute 4-axis rotations of a Job or a group of Jobs. After the 4th-axis is rotated, the job will be executed again. When you choose this option, the **Rotations list dialog** will be displayed.



Angle: 120

List of Angles

1	15
2	30
3	60

☑ Include Original

1. Enter the value of the angle (+/-) around which the 4th axis should be rotated.

2. Click **Enter** to add the angle to the list of angles.

3. Check the **Include Original** box, if you want to execute the job in its original position before rotating it.

4. Confirm with **OK**.

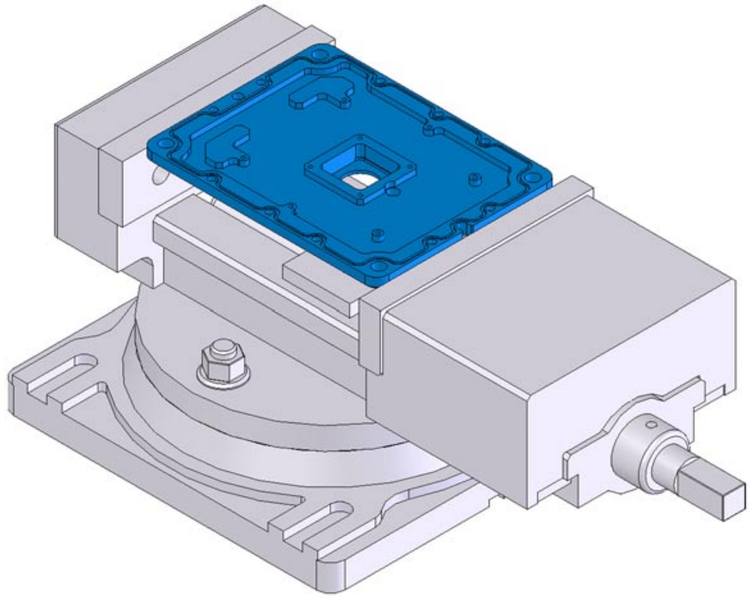
- To edit an angle in the angles list, enter the new value for this angle into the **Edit** field, click on the angle you want to overwrite and then click on **Enter**.

- To delete an angle from the angle list, click on the angle and press the **Del**-key.

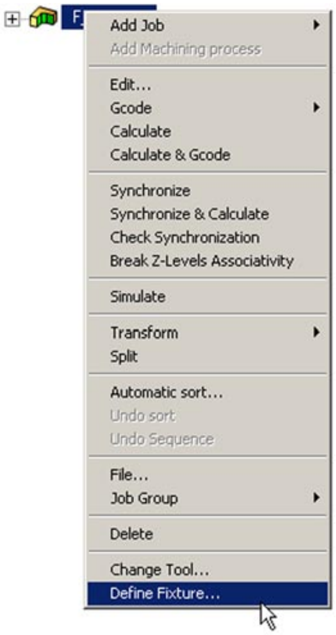
5.6 Fixture

SolidCAM enables you to define the **Part Fixtures** such as like clamps, vises, jig plates etc…

This feature enables you to get a more realistic picture during the simulation and check possible collisions between the cutting tools and fixtures.

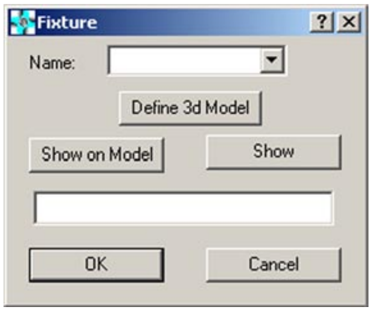


1. The **Fixture** is defined with the **Define Fixture…** item from the right-click menu available on **Jobs**.


Add Job
Add Machining process
Edit…
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort…
Undo sort
Undo Sequence
File…
Job Group
Delete
Change Tool…
Define Fixture…

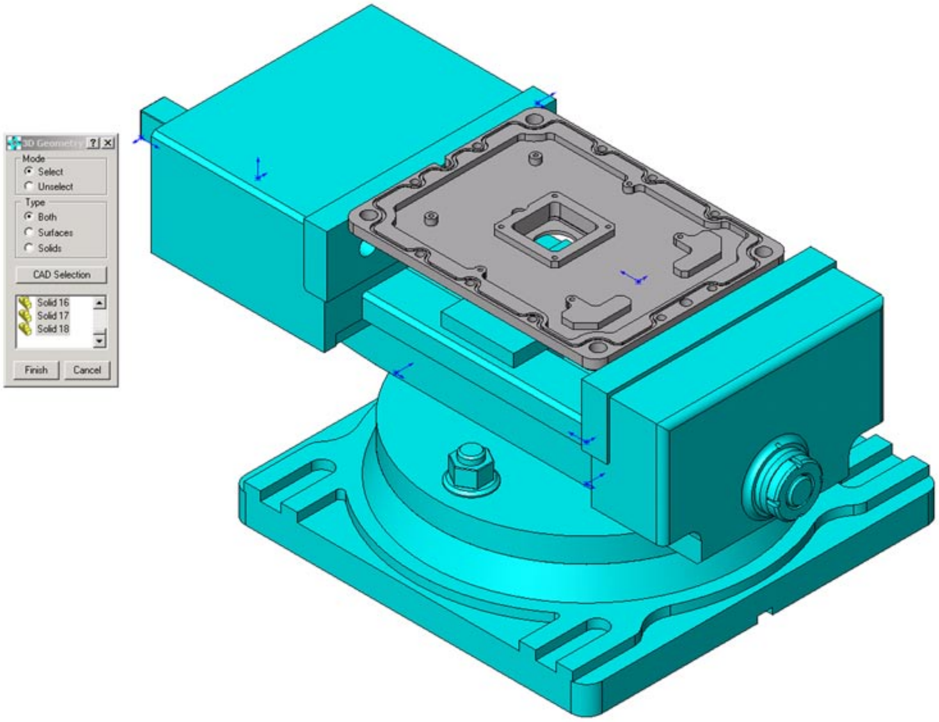
2. The **Fixture dialog** will be displayed. This dialog enables you to define the **Fixture** geometry.

5.6.1 Fixture dialog


Fixture
Name:
Define 3d Model
Show on Model
Show
OK
Cancel

- **Define 3d Model**

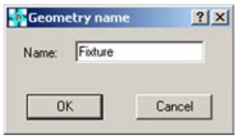
This button enables you to select the 3D model for the Fixture. The **3D Geometry dialog** will be displayed.


3D Geometry
CAD Selection
Solid 16
Solid 17
Solid 18
Finish
Cancel

Mode
● Select
○ Unselect

Type
● Both
○ Surfaces
○ Solids

When the geometry is selected, click on the **Finish** button. The **Geometry name dialog** will be displayed.

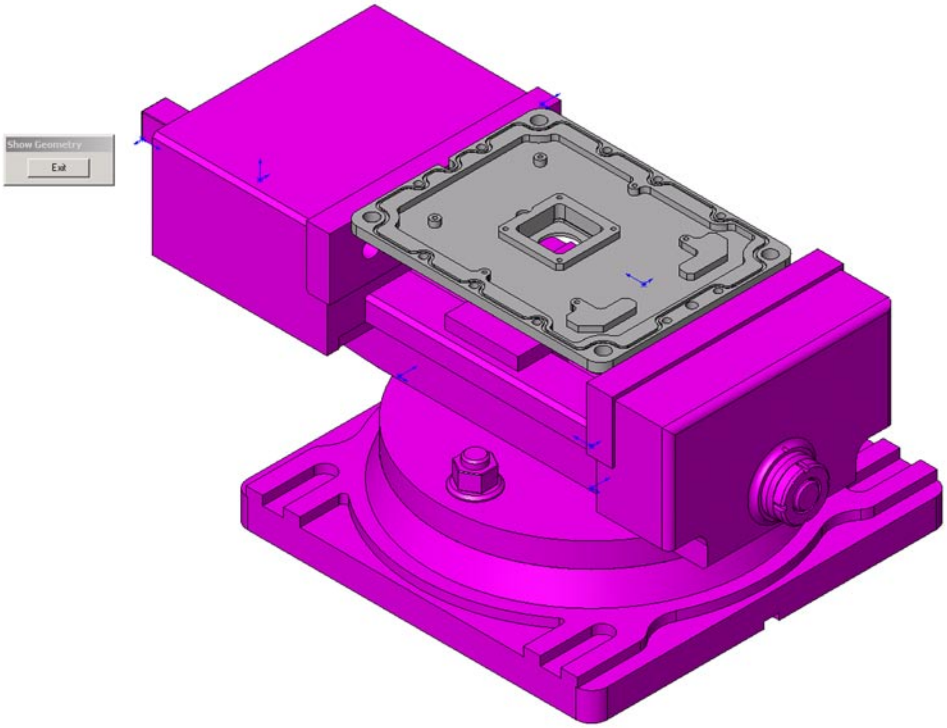

Geometry name
OK
Cancel

Name: Fixture

In this dialog, confirm the **Default Fixture name** or type a new one. Confirm this dialog. The **Fixture dialog** will be displayed again.

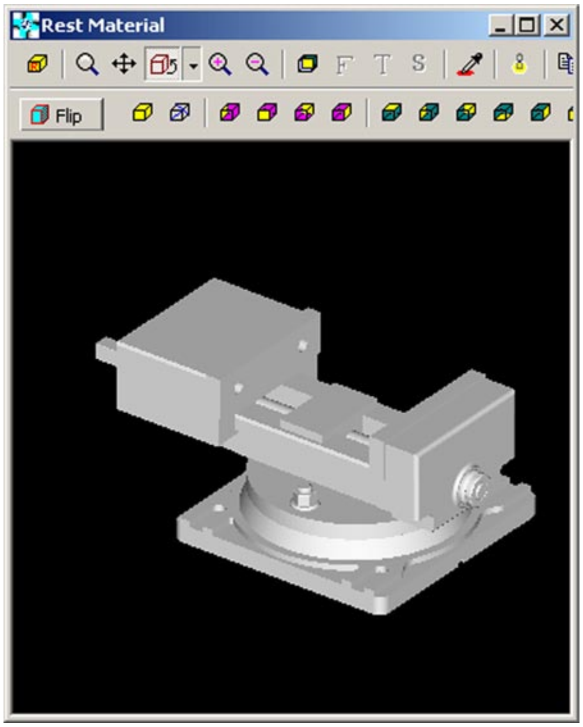
- **Show on Model**

  You can highlight the selected **Fixture** model in the **SolidWorks** window with this button.

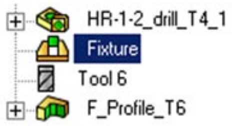

Show Geometry
Exit

- **Show**

  You can display the **Fixture** in the **Rest Material window** of SolidCAM with this button.


Rest Material
F
T
S
Flip

When the **Fixture** is defined, SolidCAM adds the **Fixture item** to the **CAM Manager**.


HR-1-2_drill_T4_1
Fixture
Tool 6
F_Profile_T6

o define any number of **Fixtures** that can be used at different stages of the

2.5D Milling

6

SolidCAM enables you to perform a number of standard 2.5 operations like profile, pocket, slot machining etc. These operations can be performed using 2D skethces as well as solid models.

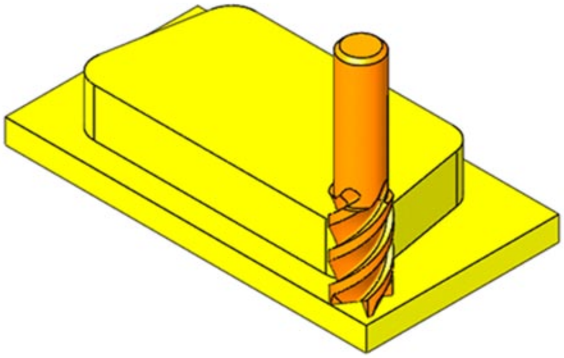
Full toolpath control and powerful algorithms ensure that the user can manufacturethe way he needs to.

- ❒ Profile Job
- ❒ Pocket Job
- ❒ Drill Job
- ❒ Slot Job
- ❒ Translated surface Job

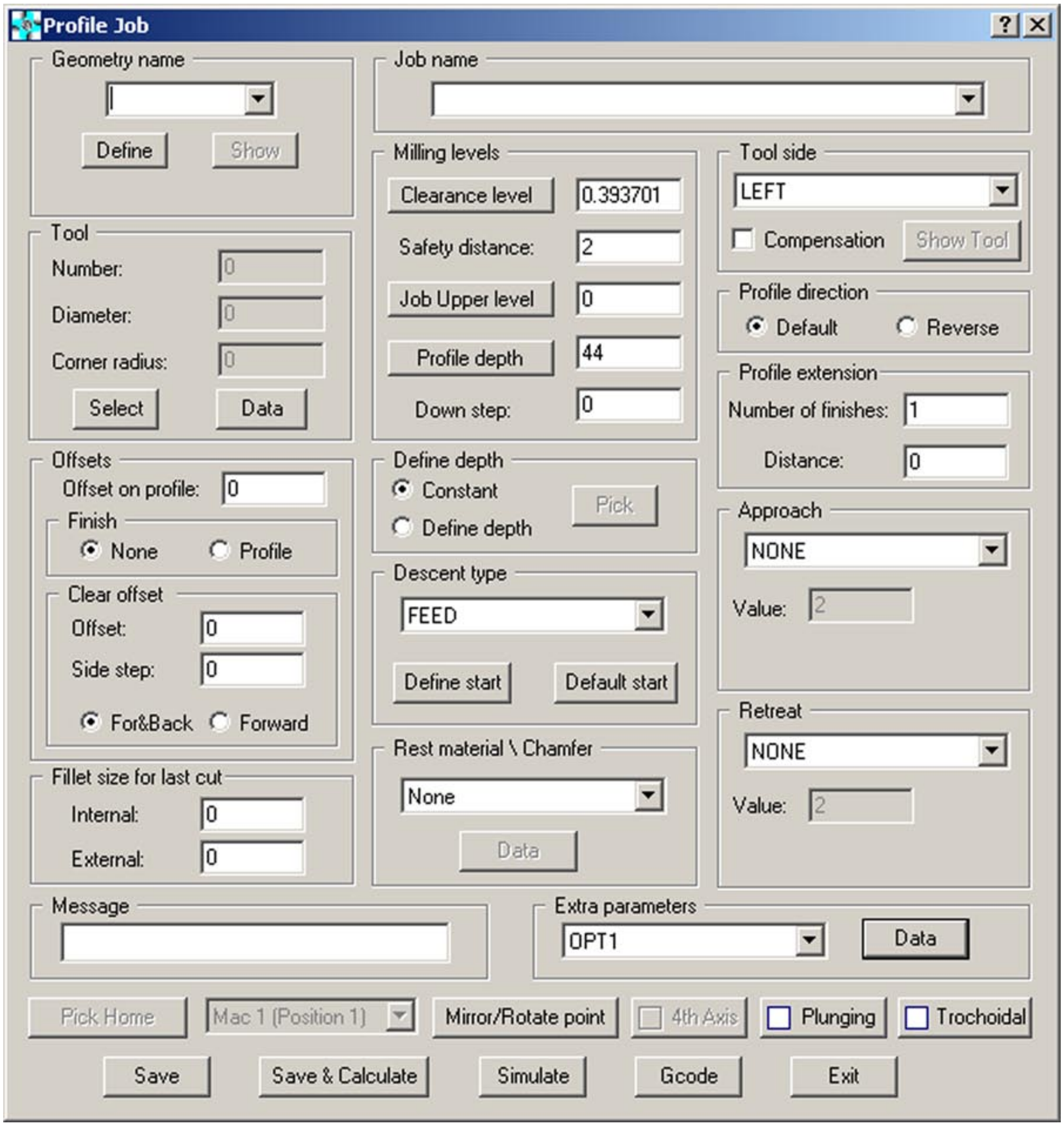
6.1 Profile Job

You can mill on or along a contour. The profile geometry can be either open or closed. In profile milling you can optionally use tool radius compensation to the right or left side of the geometry. SolidCAM offers two types of profiling:

- Milling a **single profile** to the specified constant or variable depth in one step or in several user-defined down-steps.
- **Concentric profiles** to the specified constant or variable depth; this type of profiling generates several concentric profiles that start from the defined clear offset distance from the profile, and finishes on the profile geometry, thus clearing the area around the profile to a constant depth.



When you select the commands **Add>Profile**, the **Profile Job** screen will be displayed.



Profile Job

Geometry name:

Job name:

Milling levels
Clearance level: 0.393701
Safety distance: 2
Job Upper level: 0
Profile depth: 44
Down step: 0

Tool side: LEFT
☐ Compensation

Profile direction: ● Default ○ Reverse

Profile extension
Number of finishes: 1
Distance: 0

Tool
Number: 0
Diameter: 0
Corner radius: 0

Offsets
Offset on profile: 0
Finish: ● None ○ Profile
Clear offset
Offset: 0
Side step: 0
● For&Back ○ Forward

Define depth: ● Constant ○ Define depth

Descent type: FEED

Approach: NONE
Value: 2

Retreat: NONE
Value: 2

Fillet size for last cut
Internal: 0
External: 0

Rest material \ Chamfer: None

Message:

Extra parameters: OPT1

Mac 1 (Position 1)
☐ 4th Axis ☐ Plunging ☐ Trochoidal

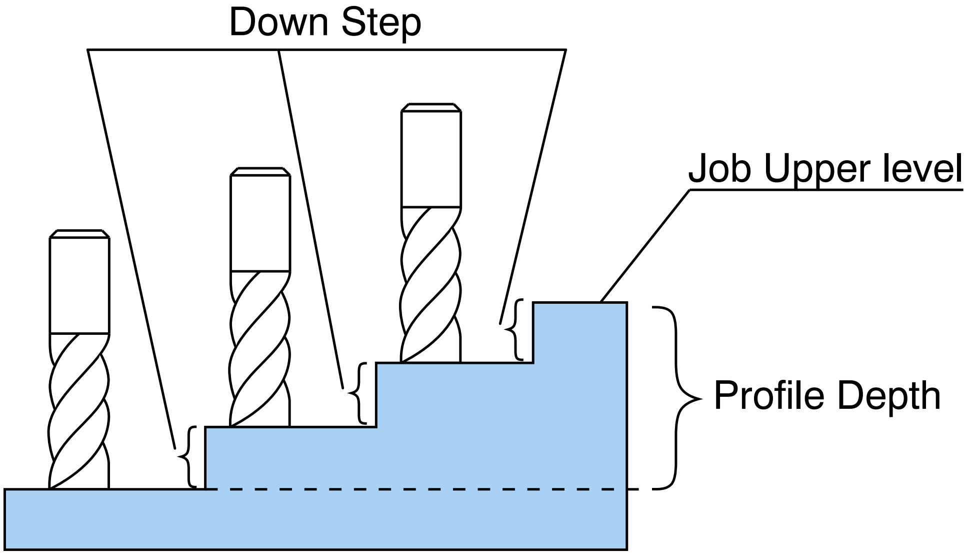
Define
Show
Select
Data
Pick
Define start
Default start
Show Tool
Data
Data
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

6.1.1 Geometry name

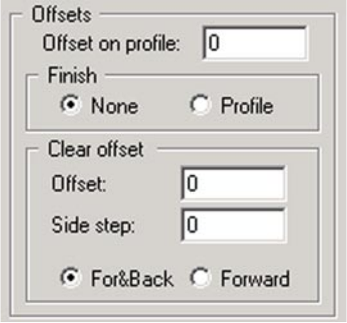
In this section you have to specify the contour, i.e. profile geometry you want to mill.

6.1.2 Down Step

SolidCAM uses the constant-Z passes for the Profile generation. With the **Down Step** parameter you can define the distance between each two successive Z-levels. Enter the tool's cutting depth.



6.1.3 Offsets



Offset on profile: 0

● None ○ Profile

Offset: 0

Side step: 0

● For&Back ○ Forward

Offset on profile

Enter an offset distance of the tool from the profile.

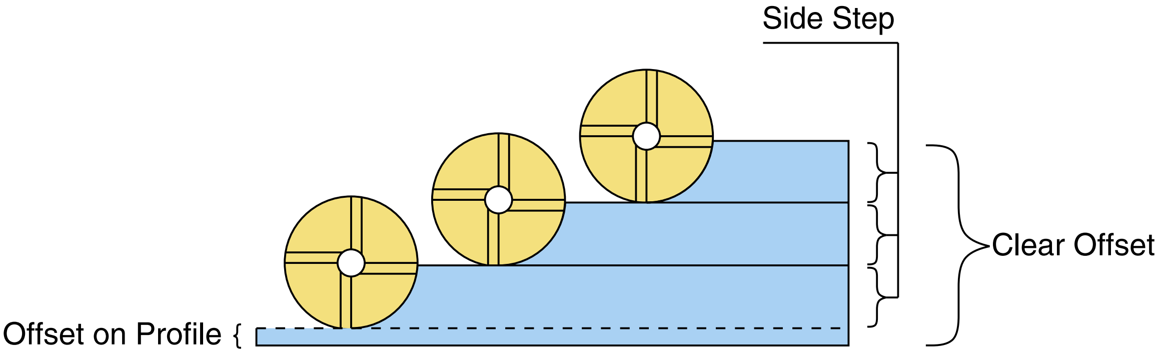
Finish offers two possible options:

- **None** - is used when no finish is required. The specified offset will remain on the contour.
- **Profile** - is used when you want to execute a finish cycle to remove the remaining offset from the profile.

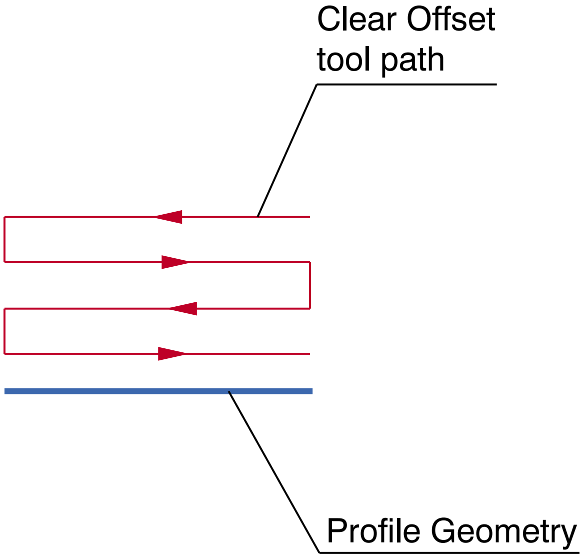
Clear offset

The Clear option will generate several concentric profiles with a constant depth that start from the defined clear offset distance from the profile and finish up to the geometry of the profile, thus clearing the area around the profile.

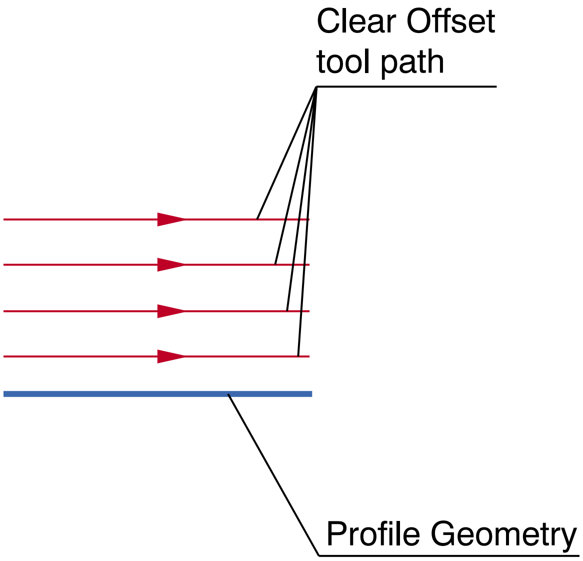
- **Offset** defines the distance from the geometry at which the milling starts. The Clear Offset value should be equal or larger than the Offset on the profile value. The tool will start milling the profile at the distance defined by Clear Offset and will finish at the distance defined by the Offset on profile; the overlap of the adjacent tool paths is defined by the Side Step.
- **Side Step** defines the overlap of adjacent tool paths. It determines the offset between two successive concentric profiles.



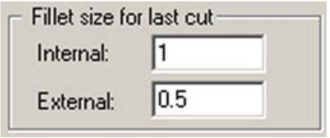
- **For&Back** – this option enables you to create the tool path for the Clear Offset removal containing both climb and conventional movements.



- **Forward** – this option enables you to create the tool path for the Clear Offset removal containing only climb movements.

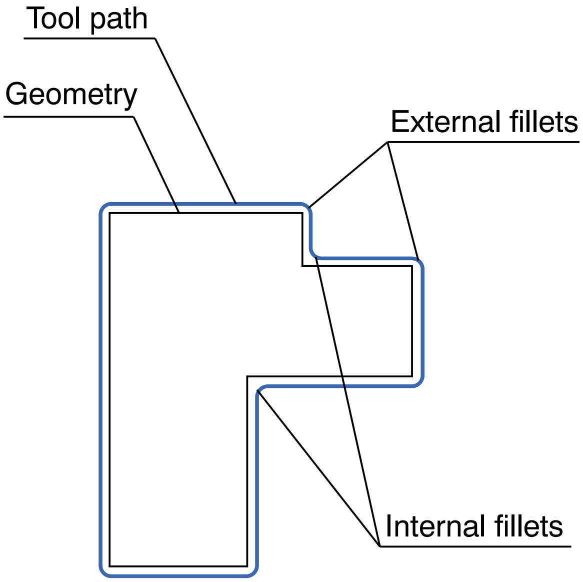

Clear Offset tool path
Profile Geometry

6.1.4 Fillet size for last cut


Fillet size for last cut
Internal: 1
External: 0.5

This option enables you to add a radius to a sharp corner without having to change the geometry.

This option is relevant for **Profile Job**, **Pocket Job**, **Slot Job (Constant section type)** and the 3D Model Job (**Roughing** and **Constant-Z finishing** strategy).


Tool path
Geometry
External fillets
Internal fillets

- **Internal**

  This field controls internal corners.

- **External**

  This field controls external corners.

- The maximum external fillet size is defined by **SolidCAM Settings** (see the topic **10.3.5**.).

  - In the **Profile Job**, this option doesn‘t work with the **Tool side: Middle** option.

6.1.5 Define depth

This option enables you to choose the depth type for the profile.

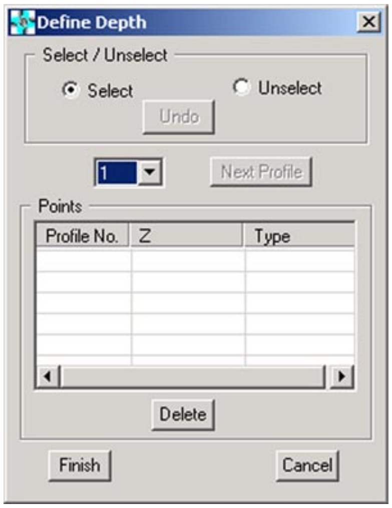
- **Constant**

  With this type SolidCAM generates a tool path with a constant depth as defined in the **Milling levels area**.

- **Variable (Define Depth)**

  With this option you can define a different depth at different profile points.

1. Choose the **Define depth** mode.
2. Click on the **Pick** button. The **Define depth dialog** will be displayed.


Define Depth
Select / Unselect
● Select
○ Unselect
Undo
1
Next Profile
Points
Profile No.
Z
Type
Delete
Finish
Cancel

With this dialog you can control the points where the depth changes.

Select/Unselect

These buttons enable you to switch between the **Select/Unselect** mode to define or remove points where the depth changes. **Undo** - this button will delete or restore the last selection.

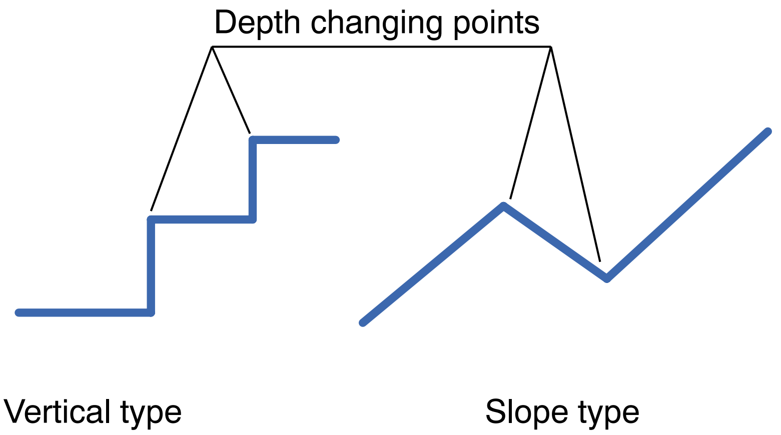
You can specify a certain chain in the profile geometry with the combo-box with the profile number and with the **Next Profile** button.

The **Points field** contains information about selected depth changing points. You can change the point's properties by double-clicking the parameter in this area.

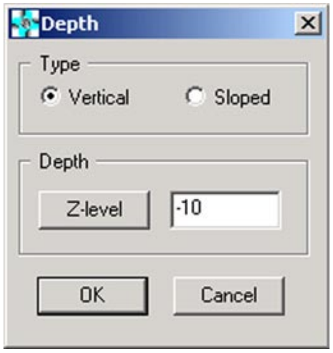
- **Profile No.** - the Profile geometry chain number.
- **Z** - the Z coordinate of the profile depth at the specified point.
- **Type** - the depth changing type.

Vertical - the depth changes at specified points.

Sloped - the depth changes gradually over the tool path segment.



Select the depth changing point on the profile. The **Depth dialog** will be shown.



Depth

Type

● Vertical ○ Sloped

Depth

Z-level -10

In this dialog specify the depth and vertical/sloped type.

6.1.6 Descent type

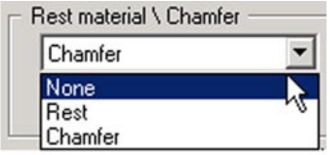
This section determines how the tool will approach the start point of the contour. Select one of the three available descent types from the list field.

- **Rapid** generates a rapid descent (G0) from the Clearance plane to the approach plane. You can use this option when you know that there is no material in the descent path.
- **Feed** will cause the tool to rapid to the Job upper plane minus the Safety distance. Then the tool will descent to the approach plane with the given Z Feed rate. Use this option when you know that there is material in the descent path.
- **Diagonal** can be activated only if the down step is equal to the profile depth. It will cause the tool to rapid from the Clearance plane to the Job upper plane + the Safety distance. Then the tool will diagonally descent to the start of the profile geometry at the normal feed rate. The angle of the diagonal is automatically generated depending on the type of approach you choose.

Define start enables you to pick the point on a closed profile where the tool should start to machine the profile. By default, the machining will start at one-quarter of the length of the first chain entity. This field is inactive if you use an open profile geometry, as the machining will then automatically start on the vertex of the first chain entity.

Default start will automatically select the start point of the profile chain geometry you have defined as the start point for the contour.

6.1.7 Rest Material/Chamfer



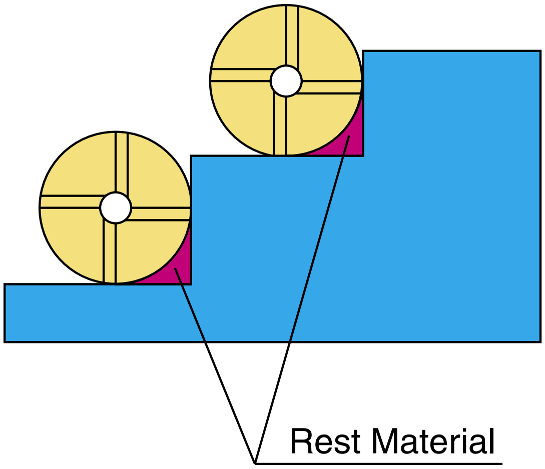
- **Rest material** - This option enables you to define parameters to remove the rest material left unmachined after previous Jobs.
- **Chamfer** - This option enables you to machine chamfers with a Center drill tool.

The **Rest Material dialog** is available from the following jobs:

- **Profile Job**
- **Pocket Job**

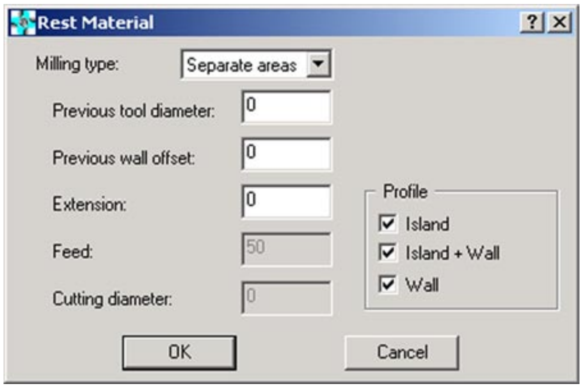
Rest material

In **Profile Machining**, when a large tool is used around the profile, the tool will leave material in corners that it cannot enter.



The rest material option enables you to remove the material from this area without defining a new geometry.

In the **Rest Material dialog** box, the following data must be given:



Rest Material

Milling type: Separate areas

Previous tool diameter: 0

Previous wall offset: 0

Extension: 0

Feed: 50

Cutting diameter: 0

Profile

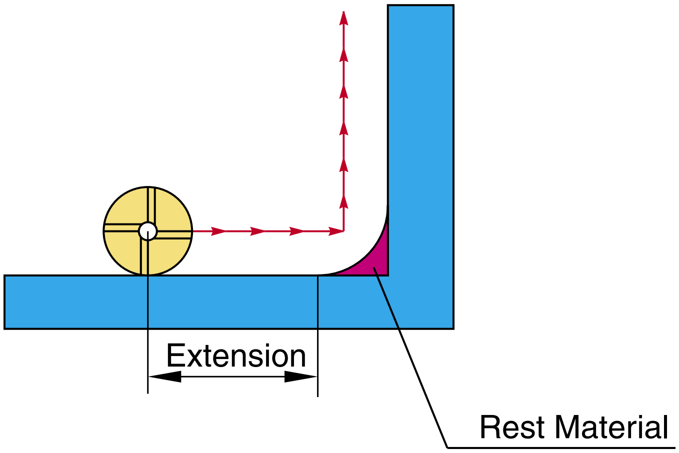
☑ Island

☑ Island + Wall

☑ Wall

In this dialog you have to define the following parameters:

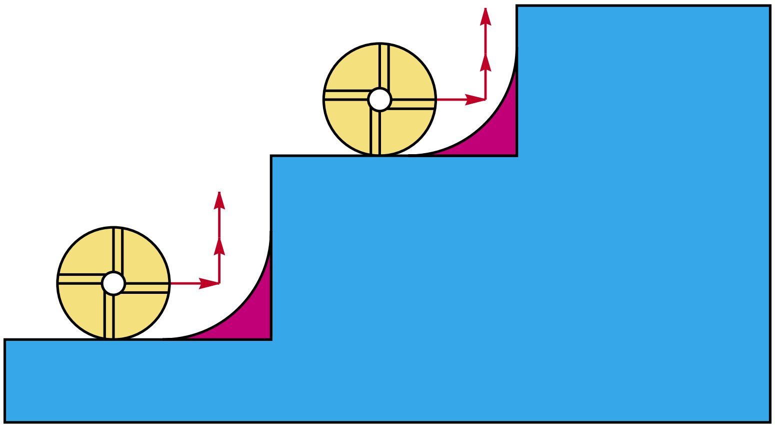
- **Previous tool diameter** - The diameter of the rough end-mill that was used in the previous job.
- **Previous wall offset** - The wall offset of the previous job.
- **Extension** - The overlap distance that you would like to start and end from the previous larger end-mill.



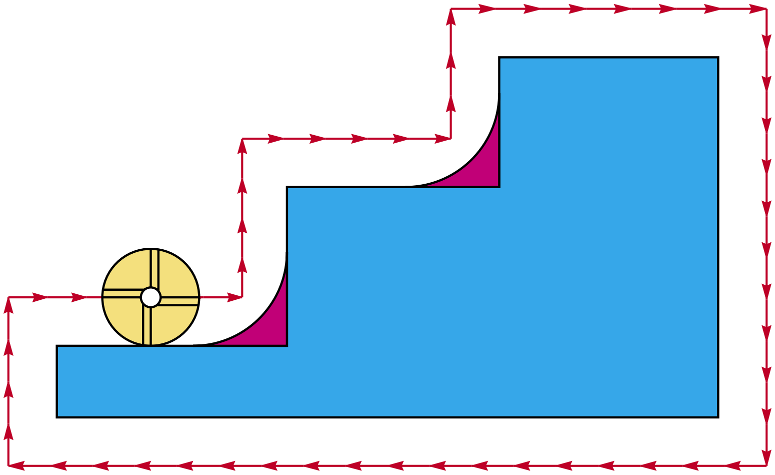
Milling type

Choose the milling type either as **Separate areas** or **Around profile**. In separate areas, SolidCAM generates a profile or pocket tool path to clean areas that the previous tool could not mill. The around profile strategy generates a closed profile to mill the rest material.

Separate areas - with this option SolidCAM machines only areas not machined with the previous tool.

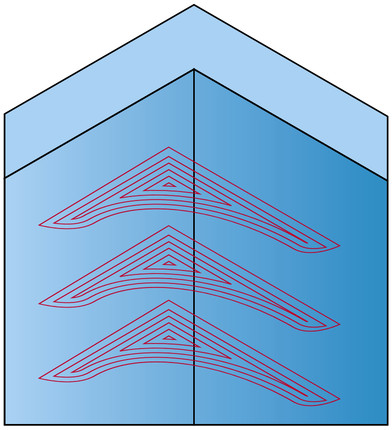


Around Profile - SolidCAM machines the whole profile.

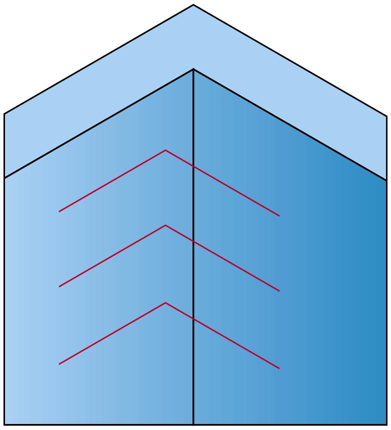


In **3D Model** and **Pocket Jobs** with **Separate Area** machining strategy, SolidCAM can generate two types of tool paths, depending on the volume of the rest material and on the previous and present tool diameters:

- **Pocket-style tool paths.**
- **Profile-style tool paths.**

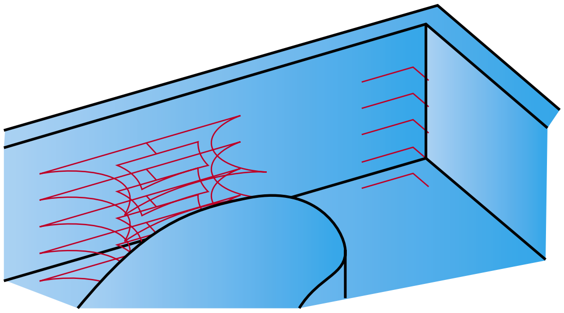


Pocket-style tool paths



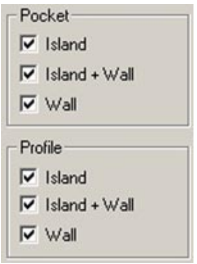
Profile-style tool paths

SolidCAM generates either a pocket or profile tool path in each area of the rest material.



The options picked in this box will determine which of these tool paths will be cut:

- **Island** – only tool paths that touch the Islands are cut.
- **Island + Wall** – tool paths that touch both the Wall and Islands are cut.
- **Wall** - only tool paths that touch the Wall are cut.



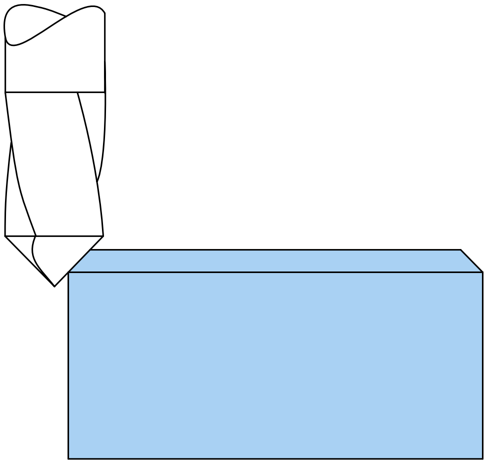
Pocket
☑ Island
☑ Island + Wall
☑ Wall

Profile
☑ Island
☑ Island + Wall
☑ Wall

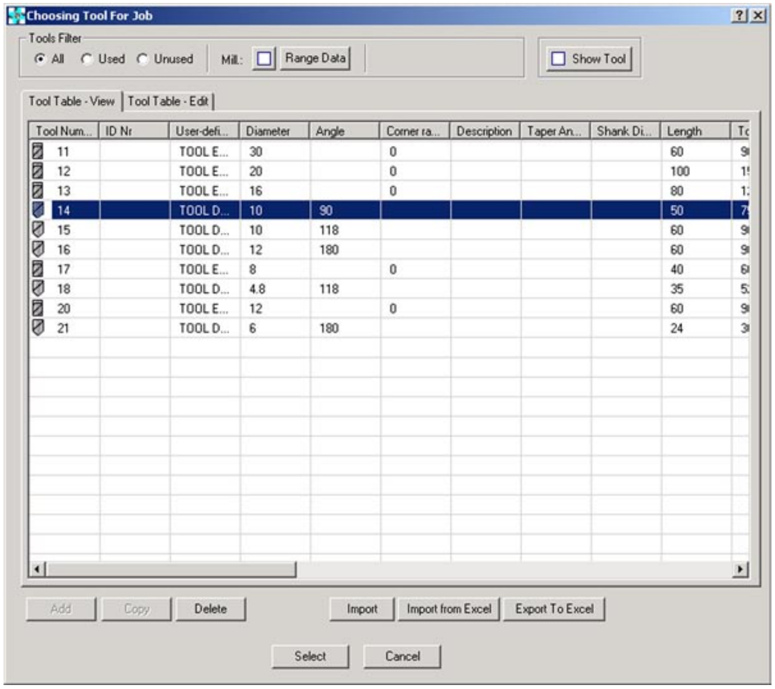
Chamfer

The **Chamfer** option is available only in the Profile and Pocket Jobs.

The chamfer feature in the profile and pocket job enables you to use a drill tool to add a chamfer to the edge of a part.

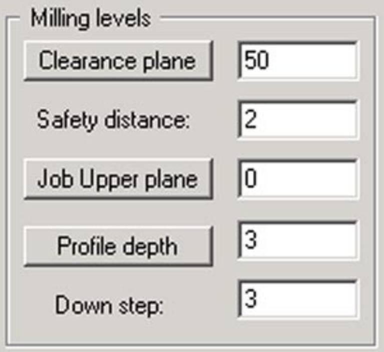


In the Tool area of the Profile Job screen, click on the **Select** button. The **Part Tool Table dialog** will be displayed from which you can choose the drill tool for the chamfering.


Choosing Tool For Job
?
×
Tools Filter
● All
○ Used
○ Unused
Mill: ☐
Range Data
☐ Show Tool
Tool Table - View
Tool Table - Edit
Add
Copy
Delete
Import
Import from Excel
Export To Excel
Select
Cancel

Tool Num...	ID Nr	User-defi...	Diameter	Angle	Corner ra...	Description	Taper An...	Shank Di...	Length	Tc
11		TOOL E...	30		0				60	9
12		TOOL E...	20		0				100	1
13		TOOL E...	16		0				80	1
14		TOOL D...	10	90					50	7
15		TOOL D...	10	118					60	9
16		TOOL D...	12	180					60	9
17		TOOL E...	8		0				40	6
18		TOOL D...	4.8	118					35	5
20		TOOL E...	12		0				60	9
21		TOOL D...	6	180					24	3

In the **Milling levels** section, enter the size of the chamfer edge in the **Profile depth** window.



Milling levels

Clearance plane: 50

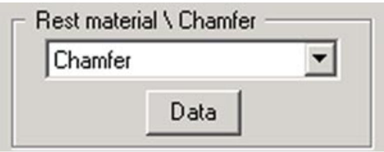
Safety distance: 2

Job Upper plane: 0

Profile depth: 3

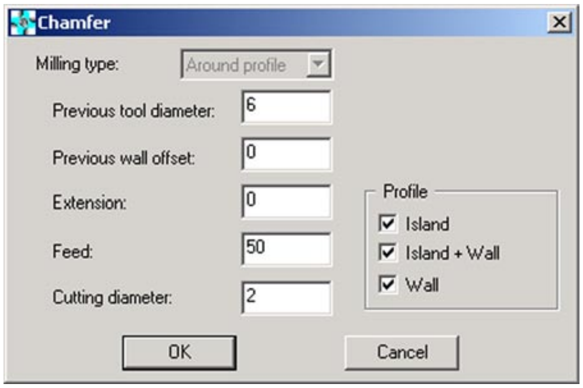
Down step: 3

Choose **Chamfer** from the **Rest Material/Chamfer** option and press the **Data** button.



Rest material \ Chamfer: Chamfer

In the dialog window that opens enter the following parameters:



Chamfer

Milling type: Around profile

Previous tool diameter: 6

Previous wall offset: 0

Extension: 0

Feed: 50

Cutting diameter: 2

Profile

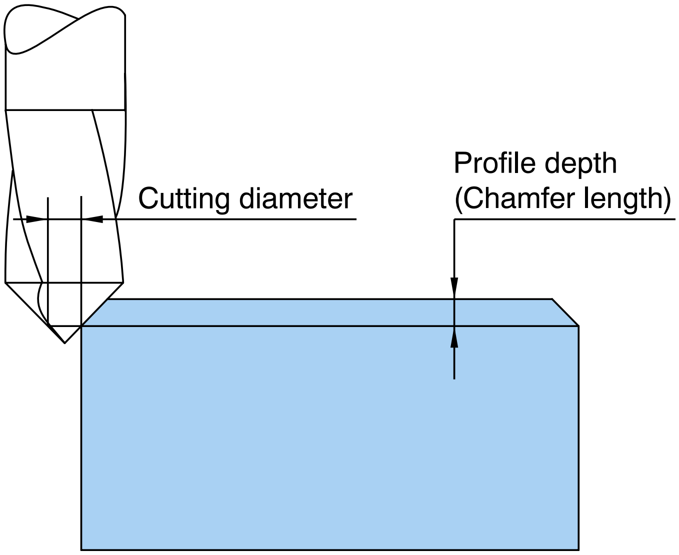
☑ Island

☑ Island + Wall

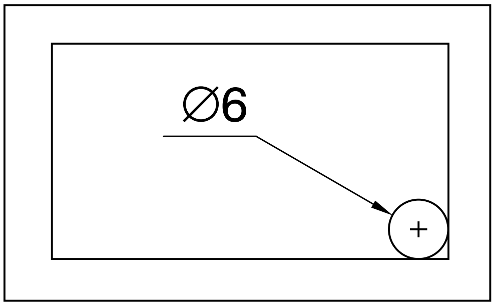
☑ Wall

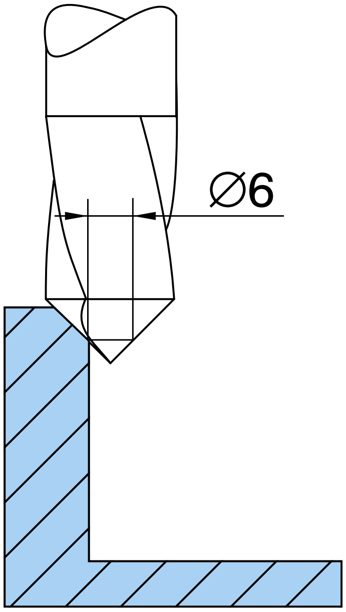
Feed - The feed rate you would like the tool to travel

Cutting Diameter - The starting cutting diameter of the tool



If you choose to chamfer a part that has a sharp corner, it is important that the cutting diameter is the same as the last tool used in that geometry. This will give you an equal chamfer all around.

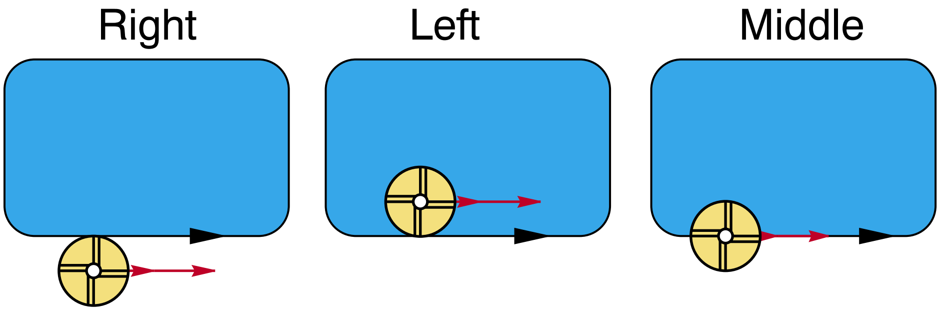




6.1.8 Tool Side

The direction of the geometry can be controlled by the chain direction and the options you use in the Profile Direction field.

Tool Side enables you to decide on the tool position relative to the geometry.



- **Right** - The tool will cut on the right side of the profile geometry at a distance defined by the values set in **Offset on profile** (see the topic **6.1.3**).
- **Left** - The tool will cut on the left side of the profile geometry at a distance defined by the values set in **Offset on profile** (see the topic **6.1.3**).
- **Middle** - The center of the tool will move on the profile geometry. (NO compensation G4x can be used with this option).

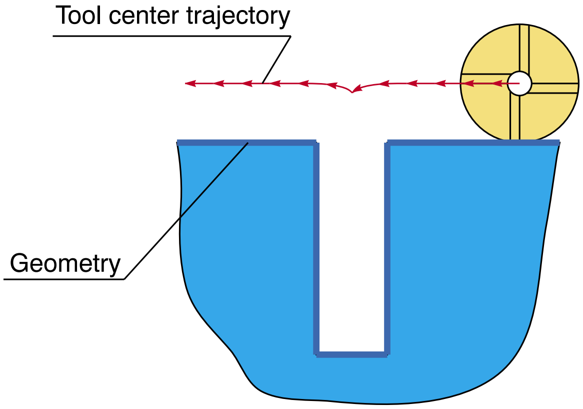
- **Show** will display the chain geometry of the profile, its direction and a circle representing the tool relative to the geometry. The display uses the current settings in the Profile direction and Tool side fields.

6.1.9 Compensation

If **Compensation** is activated, the tool radius compensation options G4x of the CNC controller are used in the G-Code.

No compensation

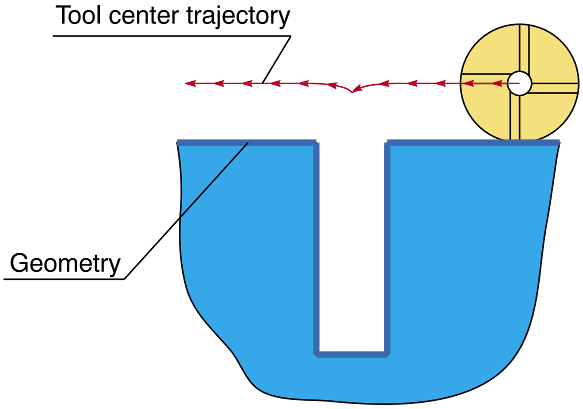
When the compensation is turned off, SolidCAM generates a trajectory of the tool center as an offset by the tool radius from the geometry.



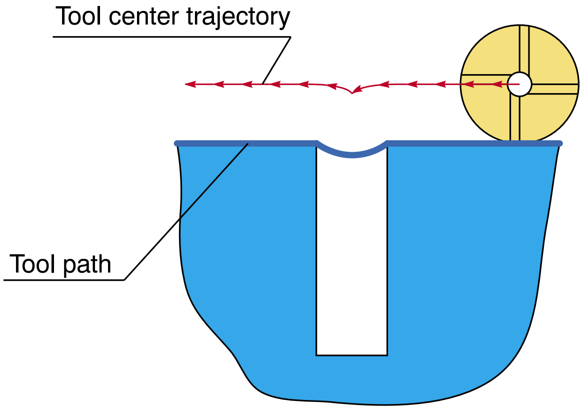
This tool center trajectory will be performed in the G-code. No compensation will be used.

Radius compensation

When the compensation is turned on, SolidCAM generates the trajectory of the tool center as an offset by the tool radius from the geometry.



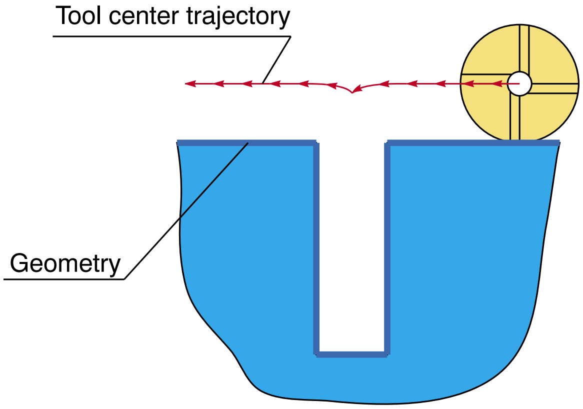
At the next stage SolidCAM creates the tool path as the offset by the tool radius from the tool center trajectory. The offset will be created in the geometry direction.



The G-code will be created for the tool path. The radius compensation will be used in the G-code. The **Offset Amount** will be equal to the Tool Radius.

Zero Compensation

With this method, SolidCAM calculates the trajectory of the tool center as the offset from the geometry.



This trajectory will be used in the G-code. The radius compensation will be used. The **Offset Amount** will be equal to **0**.

Settings

SolidCAM enables you to define the type of compensation that will be used by the specific variable in the MAC-file.

Comp_by_zero_tool = N – In this case the Radius compensation will be used.

Comp_by_zero_tool = Y – The Zero compensation will be used.

6.1.10 Profile Direction

This option controls the direction of the geometry for this particular job.

- **Default** will machine the profile in the direction the chain has been defined.
- **Reverse** changes the direction of the geometry for this job.

6.1.11 Profile extension

This option enables you to extend the profile in the positive profile direction.

- **Number of finishes** - SolidCAM enables you to generate a number of finish paths to achieve the best surface quality. You can define the number of finish paths.
- **Distance** - the extension value.

6.1.12 Approach/Retreat

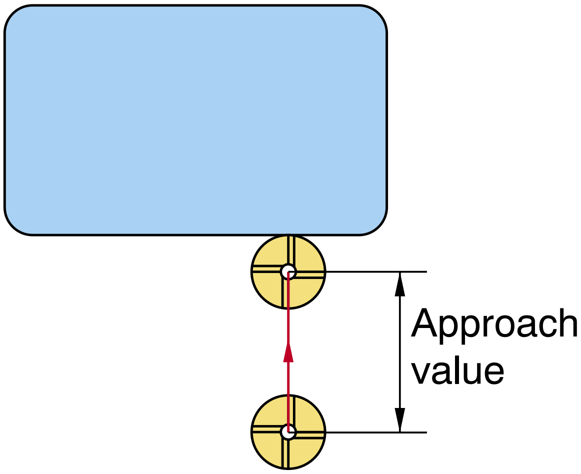
You can control the way the tool approaches and retreats from the profile. Select an option from the list field and enter the **Distance value** for this option.

None

The tool approaches/retreats to the milling level exactly adjacent to the start point of the profile.

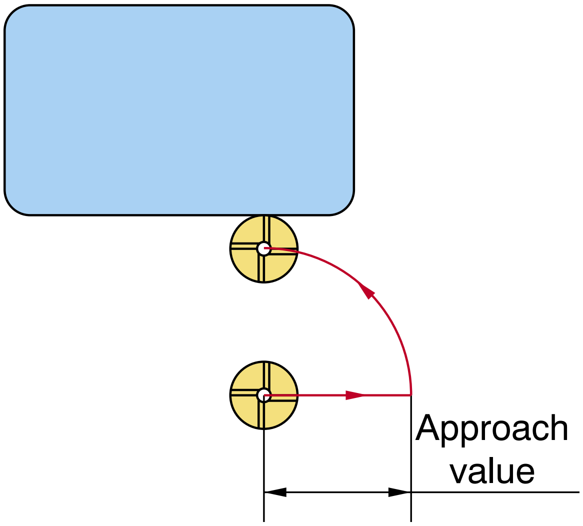
Normal

The tool approaches/retreats to the profile from a point normal to the profile. The length of the normal can be set in the **Value** field.

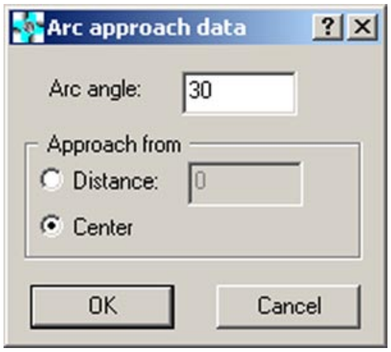


Arc

The tool approaches/retreats to the profile with a tangential arc. The arc radius can be set in the **Value** field.



The **Data** button will be displayed. Click on the button to display the **Arc approach data dialog**.



Arc angle: 30

Approach from

○ Distance: 0

● Center

Arc angle

This parameter defines the angle of the approach arc segment. The default angle value is 90°; in this case SolidCAM generates an approach path of quarter arc.

Approach from

With the Arc approach option the tool moves normal to the approach arc at the arc start-point.

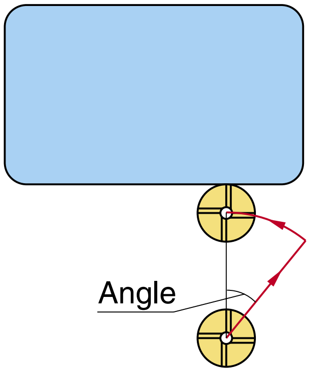
The following options are available:

Distance

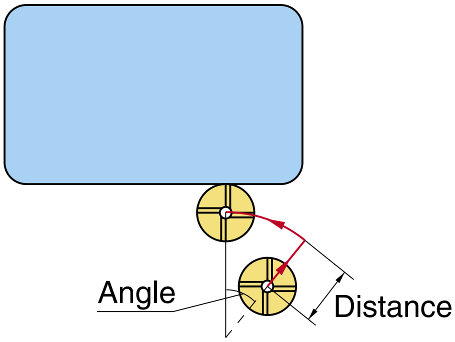
In this case the tool starts from the specified distance from the arc start-point.

Center

In this case the tool starts from the approach arc center.



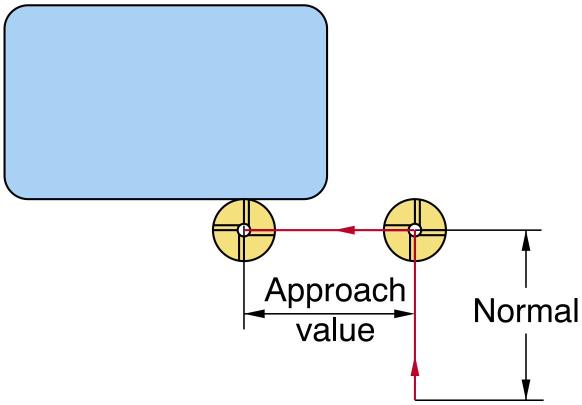
Approach from **Center**



Approach from **Distance**

Tangent

The tool approaches/retreats on a line tangent to the profile. The length of the tangent can be set in the **Value** field.

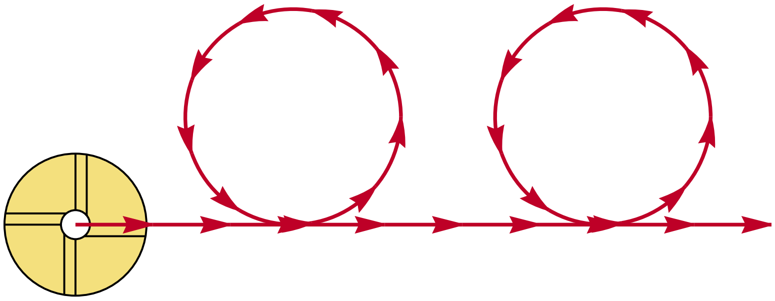


Point

The tool approaches/retreats from a user-defined position. From this position the tool will move in a straight line to the start point of the profile. When you select this option, the **Pick** button will be activated and you can select a position from the CAD model.

6.1.13 Trochoidal milling

This option enables SolidCAM to replace the straight lines in a toolpath by circular moves with a constant radius. The **Trochoidal milling** button is available in the **Profile Job** dialog.

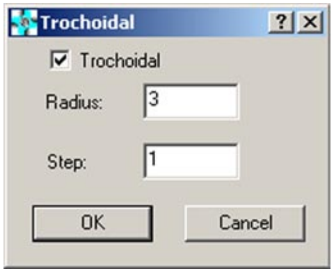


Potential benefits include:

- **Faster metal removal**. Making the constant circular curve the only direction change enables a high feed rate to be consistently maintained.
- **Longer tool life**. The cutting edge is in contact with the material through only about 5% of the cutter's revolution vs. up to 50% for more conventional cutting. Cooling of the tool is improved.

1. Click on the **Trochoidal** button.

   The **Trochoidal data dialog** will be displayed.



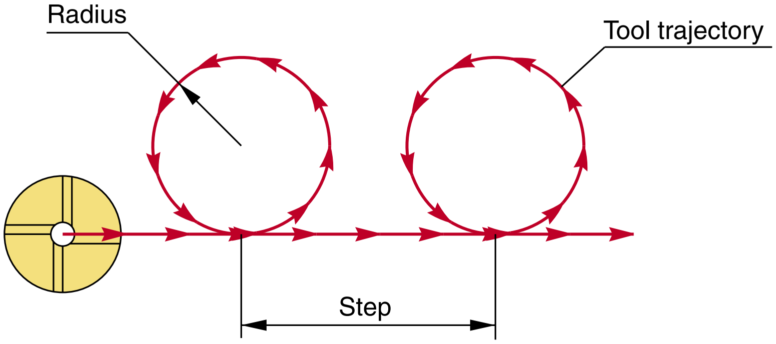
Trochoidal

☑ Trochoidal

Radius: 3

Step: 1

2. Choose the **Trochoidal** checkbox to activate the trochoidal milling option.
3. Set the **Radius** and **Step** of trochoidal movements



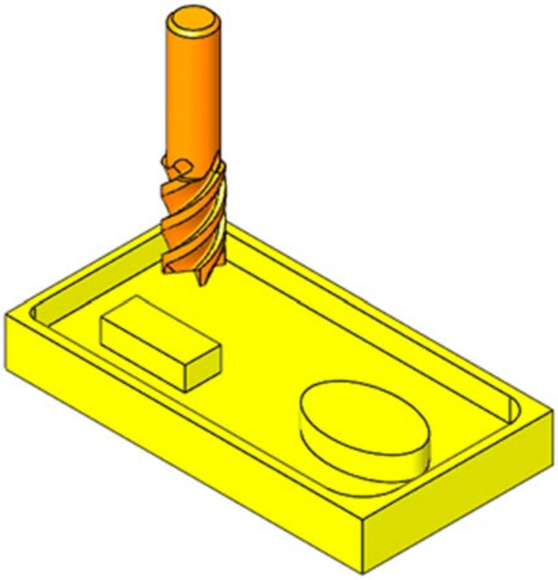
4. Confirm the data with the **OK** button.

6.2 Pocket Job

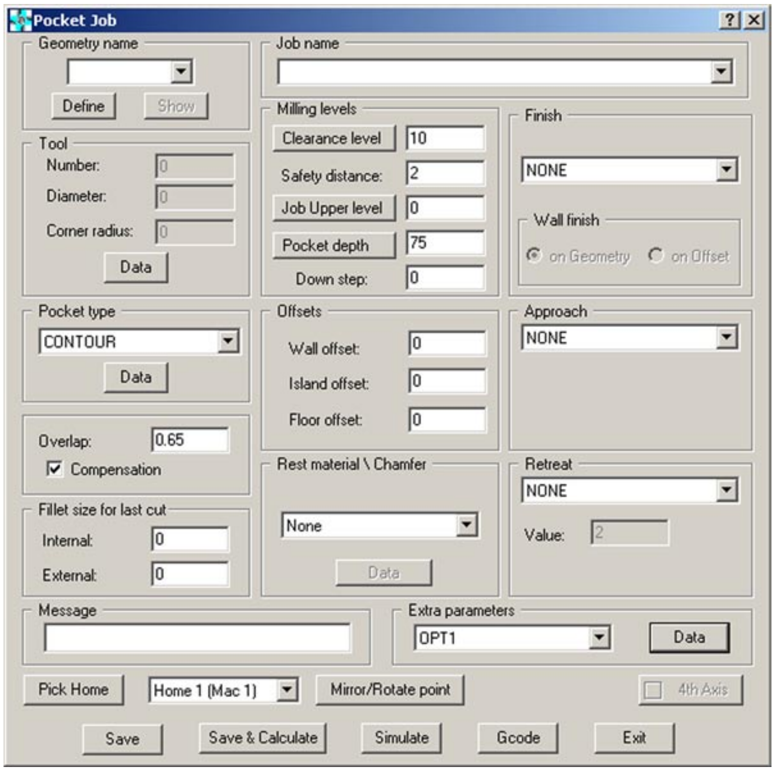
In pocket milling you have to remove material from the interior of a closed geometry. SolidCAM offers two types of pocketing:

1. When a profile geometry consists of one or more profiles and none of them are enclosed or intersect with one another, each is milled as a separate **pocket without islands**.

2. When a profile geometry consists of several profiles, any profile that is enclosed or intersected with another profile is treated as an island. You can define an unlimited number of islands within a single pocket.



When you select the commands **Add>Pocket**, the **Pocket Job** screen will be displayed.


Pocket Job
Geometry name
Define
Show
Job name
Tool
Data
Pocket type
Milling levels
Clearance level
Job Upper level
Pocket depth
Offsets
Rest material \ Chamfer
Finish
Wall finish
Approach
Retreat
Fillet size for last cut
Message
Extra parameters
Data
Pick Home
Mirror/Rotate point
4th Axis
Save
Save & Calculate
Simulate
Gcode
Exit

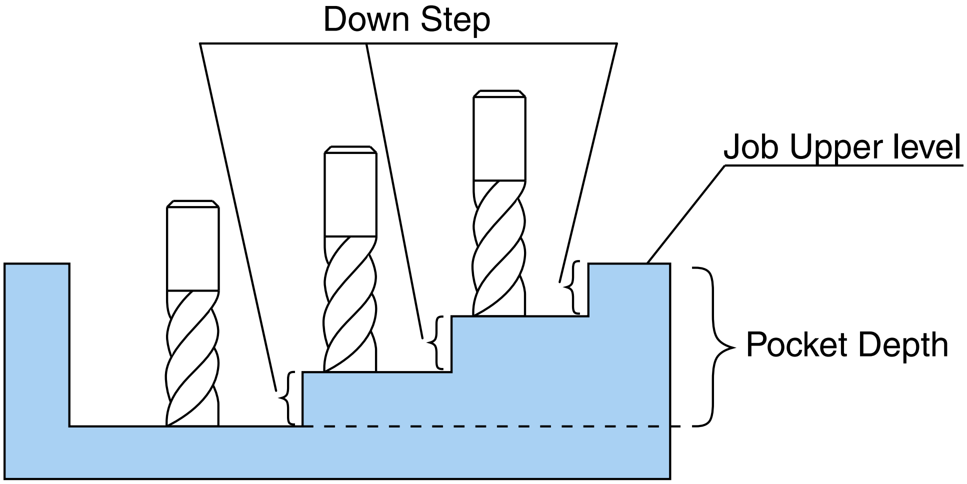
Number: 0
Diameter: 0
Corner radius: 0
Pocket type: CONTOUR
Overlap: 0.65
☑ Compensation
Internal: 0
External: 0
Clearance level: 10
Safety distance: 2
Job Upper level: 0
Pocket depth: 75
Down step: 0
Wall offset: 0
Island offset: 0
Floor offset: 0
Rest material \ Chamfer: None
Finish: NONE
Wall finish: ● on Geometry ○ on Offset
Approach: NONE
Retreat: NONE
Value: 2
Extra parameters: OPT1
Home 1 (Mac 1)
☐ 4th Axis

6.2.1 Geometry name

In this section, specify which contour, i.e. pocket geometry you want to mill.

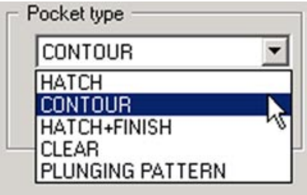
6.2.2 Down Step

SolidCAM uses the constant-Z passes for the Profile generation. The **Down Step** parameter enables you to define the distance between each two successive Z-levels. Enter the tool's cutting depth.

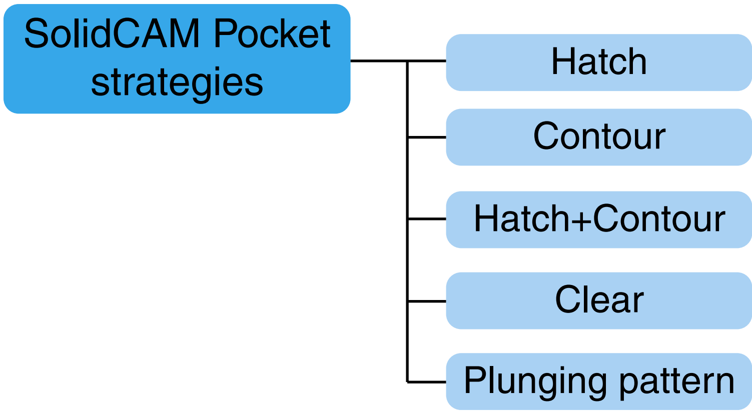

Down Step
Job Upper level
Pocket Depth

6.2.3 Pocket Types

Use the list field to select one of 5 different **Pocket types**.


Pocket type
CONTOUR
HATCH
CONTOUR
HATCH+FINISH
CLEAR
PLUNGING PATTERN

Choose the pocket type and click on the **Data** button to define additional parameters.


SolidCAM Pocket strategies
Hatch
Contour
Hatch+Contour
Clear
Plunging pattern

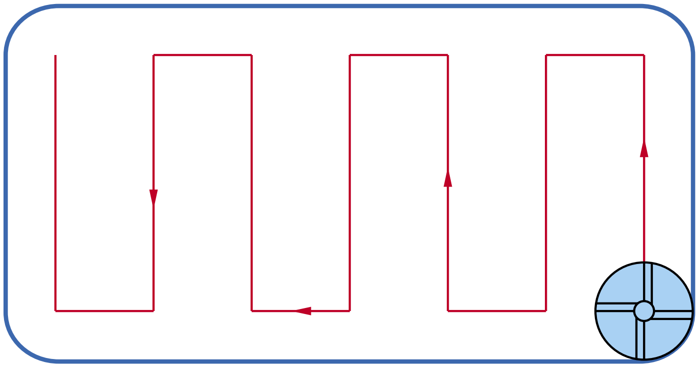
- **Contour** machines the pocket in a round pattern.
- **Hatch** mills the pocket in a linear pattern.
- **Hatch+Finish** mills the pocket in a linear pattern and cleans up the profile on each cutting depth.
- **Clear** machines an area that is half a tool diameter larger than the pocket geometry in a hatch-style linear pattern. This enables you to clean the upper face of a workpiece.
- **Plunging pattern** - the tool moves up and down in a drilling motion, traveling inside the pocket.

6.2.4 Hatch type

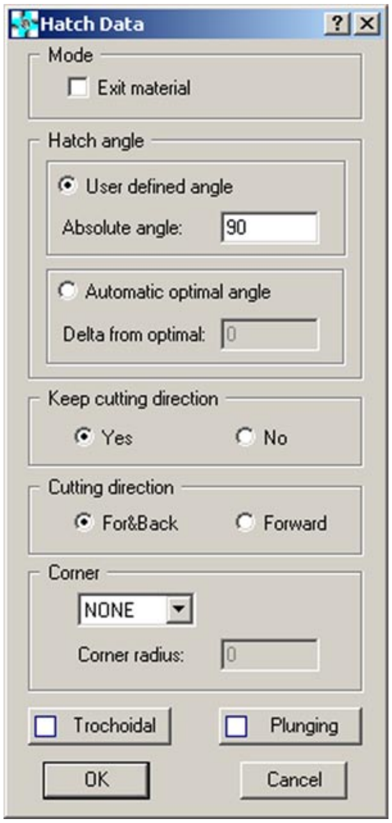
This strategy enables you to machine the pocket in a linear pattern.

This option is used in **Pocket Job** and in **3D model Job** for **Roughing** (see the topic **7.4.**) and **Constant-Z** finishing (see the topic **7.5.8.**).

Hatch type will create a linear raster tool path upto the pocket contour.



Click on the **Data** button. The **Hatch Data dialog** will be displayed.



Hatch Data

Mode

☐ Exit material

Hatch angle

● User defined angle

Absolute angle: 90

○ Automatic optimal angle

Delta from optimal: 0

Keep cutting direction

● Yes ○ No

Cutting direction

● For&Back ○ Forward

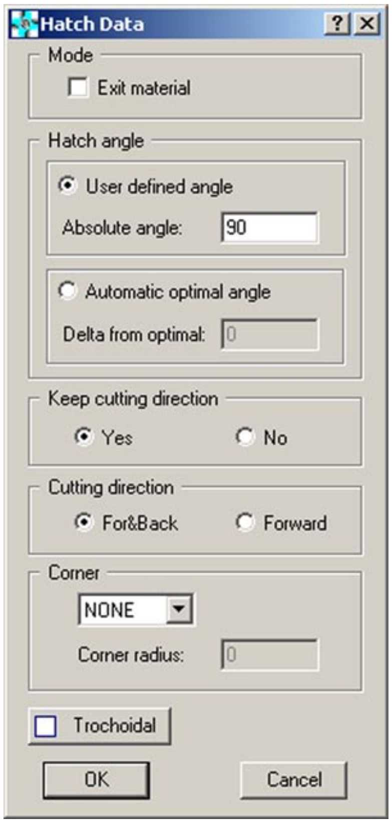
Corner

NONE

Corner radius: 0

☐ Trochoidal ☐ Plunging

Pocket Hatch Parameters



Hatch Data

Mode

☐ Exit material

Hatch angle

● User defined angle

Absolute angle: 90

○ Automatic optimal angle

Delta from optimal: 0

Keep cutting direction

● Yes ○ No

Cutting direction

● For&Back ○ Forward

Corner

NONE

Corner radius: 0

☐ Trochoidal

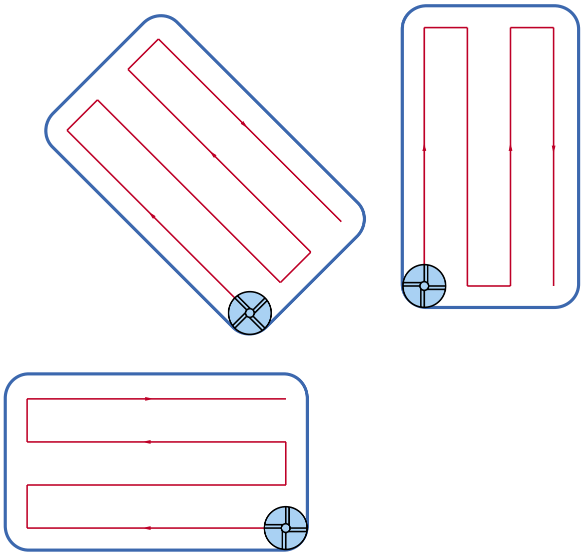
Rough & Constant-Z Hatch parameters

Hatch angle

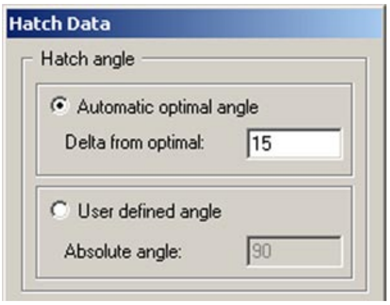
SolidCAM enables you to choose one of the following options to define the angle of hatching:

- **Automatic optional angle**

  In this case SolidCAM automatically calculates the optimal angle of hatching to expedite the machining. The tool path will always follow the length of the pocket no matter what angle the pocket is facing.



In the Automatic optimal angle option, you can change the angle by entering a different angle value in the **Delta from optimal** option.



Hatch Data

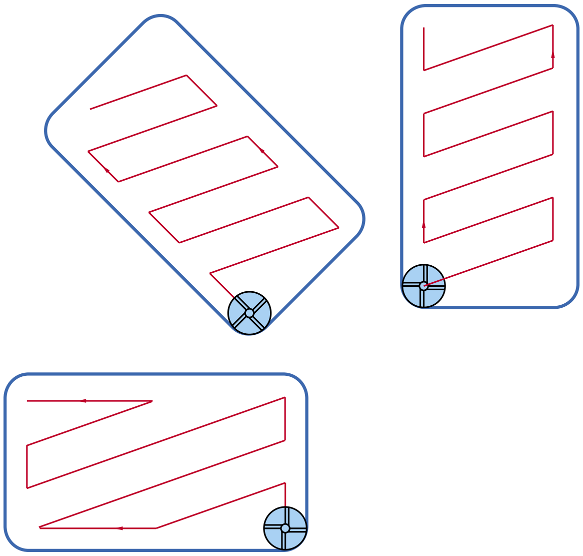
Hatch angle

● Automatic optimal angle

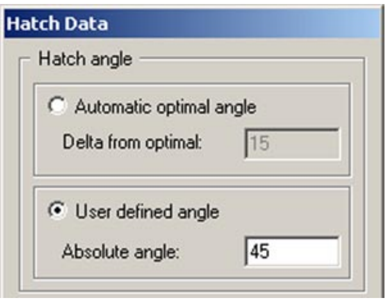
Delta from optimal: 15

○ User defined angle

Absolute angle: 90



- **User Defined angle**



Hatch Data

Hatch angle

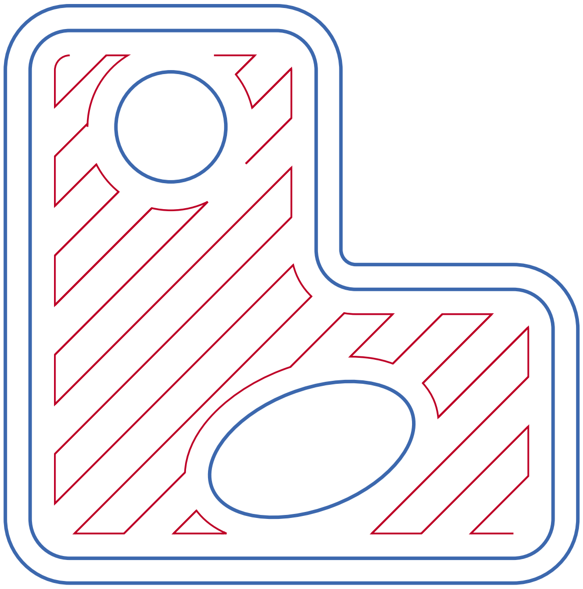
○ Automatic optimal angle

Delta from optimal: 15

● User defined angle

Absolute angle: 45

This option enables you to determine the angle of the tool path.



Keep cutting direction

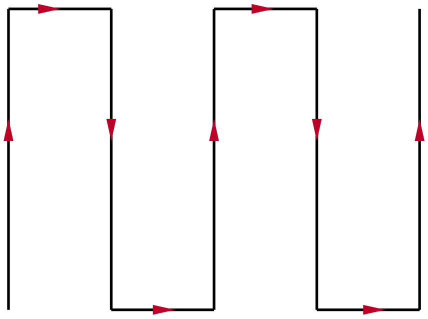
This option enables you to control the cutting direction when moving around model contours.

No. SolidCAM generates a shorter, optimized tool path by reversing some tool movements. The milling direction may change between climb and conventional milling.

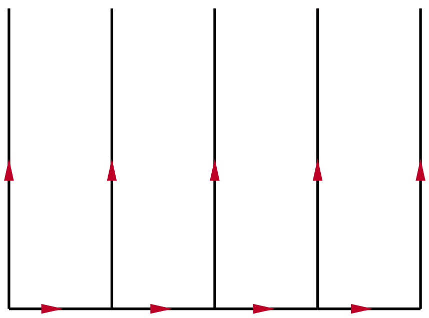
Yes. The length of the tool path will not be optimized.

Cutting direction

- **Forward** – this option enables you to create the tool path containing only one-directional movements.
- **For&Back** – this option enables you to create the tool path containing bi-directional movements.



For&Back

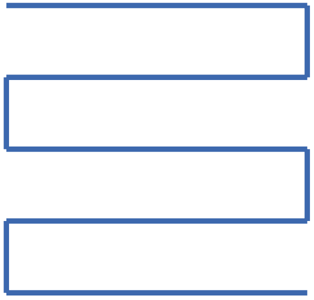


Forward

Corner

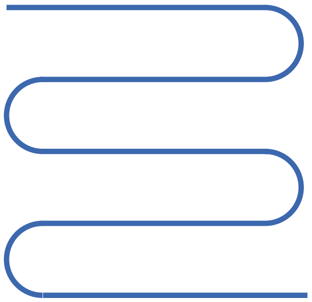
- **None**

The tool path will be formed with sharp corners between one direction and the next.



- **Fillet Corner**

Choosing Fillet will connect each direction with a given radius allowing for a smoother transition between path directions.

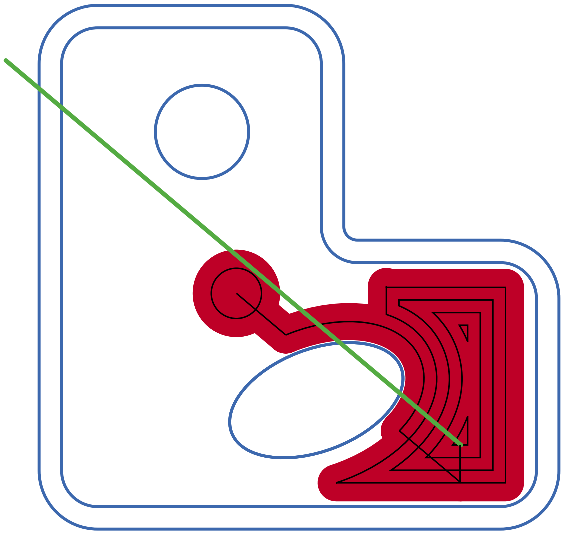


Exit Material

This option controls tool movements between the working areas.

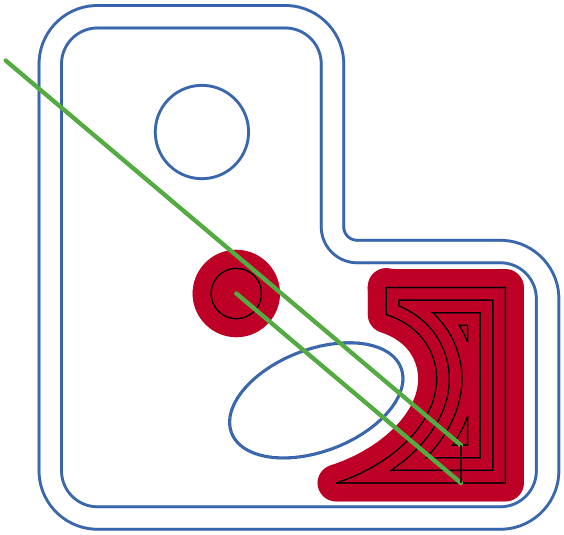
This option is used in the **Hatch** and **Contour** strategies in the **Pocket Job** and the **3D model Job**.

- **Exit material option not chosen**



When the tool moves from one working area to the next, it moves through the full material around the island to get to the next working area as shown above.

- **Exit material option chosen**



When the **Exit Material** option is chosen, the tool exits the material and travels rapidly above the material to the next working area as shown above. The approach path will be the approach you chose.

Plunging

This option is relevant only for the **Hatch strategy** in the **Pocket Job**.

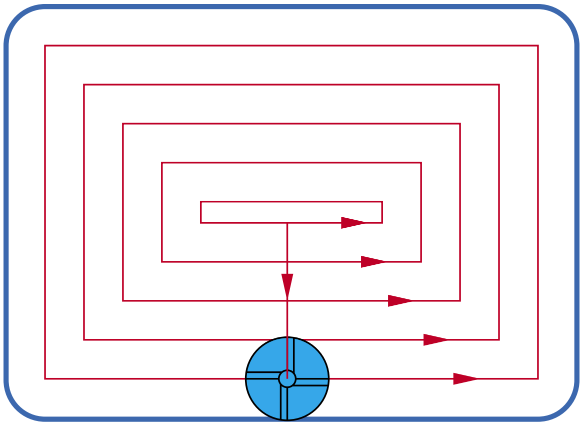
With this option SolidCAM enables you to combine the traditional hatch strategy with modern plunging technology. The **Plunging dialog** will be displayed (see the topic 6.2.8).

Trochoidal Milling

This strategy enables you to change straight tool movements with trochoidal movements. Click on the **Trochoidal** button. The **Trochoidal data dialog** (see the topic **6.1.13**.) will be displayed to specify the necessary parameters.

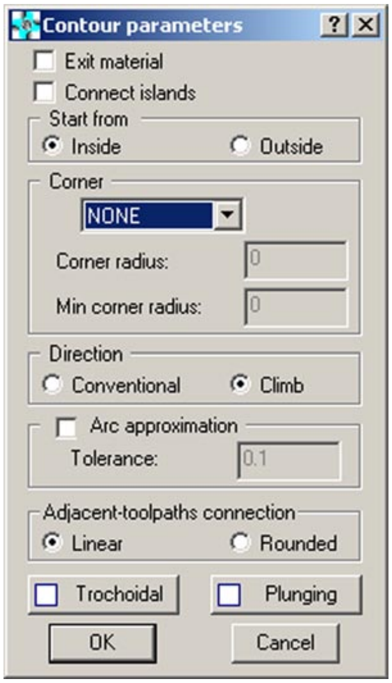
6.2.5 Contour type

With the **Contour** strategy, the tool moves on parallel offsets to the pocket contour.



Click on the **Data** button. The **Contour Data dialog** will be displayed.

This option is used in the **Pocket Job** and in the **3D model Job** for **Roughing** and **Constant-Z** finishing.



Contour parameters

☐ Exit material

☐ Connect islands

Start from: ● Inside ○ Outside

Corner: NONE

Corner radius: 0

Min corner radius: 0

Direction: ○ Conventional ● Climb

☐ Arc approximation

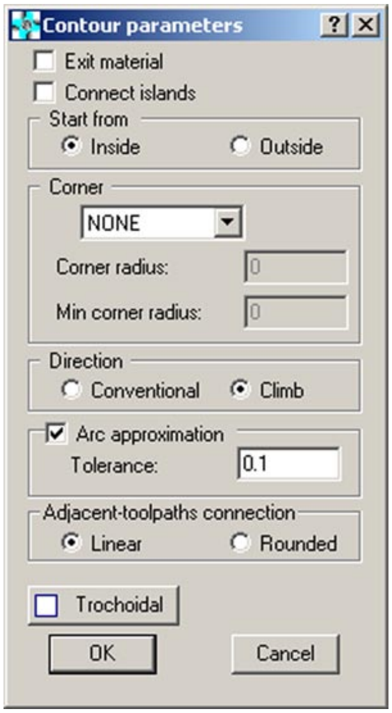
Tolerance: 0.1

Adjacent-toolpaths connection: ● Linear ○ Rounded

☐ Trochoidal ☐ Plunging

OK Cancel

Pocket Contour Parameters



Contour parameters

☐ Exit material

☐ Connect islands

Start from: ● Inside ○ Outside

Corner: NONE

Corner radius: 0

Min corner radius: 0

Direction: ○ Conventional ● Climb

☑ Arc approximation

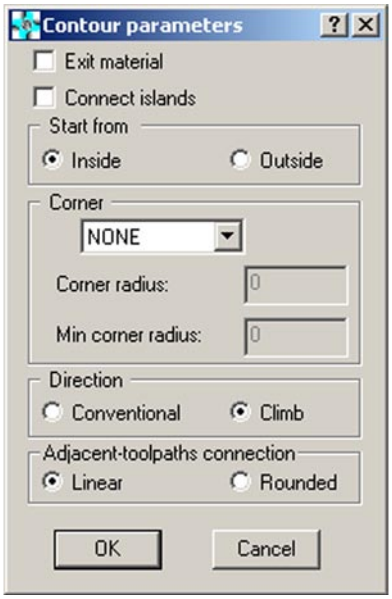
Tolerance: 0.1

Adjacent-toolpaths connection: ● Linear ○ Rounded

☐ Trochoidal

OK Cancel

Rough Contour parameters



Contour parameters

☐ Exit material

☐ Connect islands

Start from: ● Inside ○ Outside

Corner: NONE

Corner radius: 0

Min corner radius: 0

Direction: ○ Conventional ● Climb

Adjacent-toolpaths connection: ● Linear ○ Rounded

OK Cancel

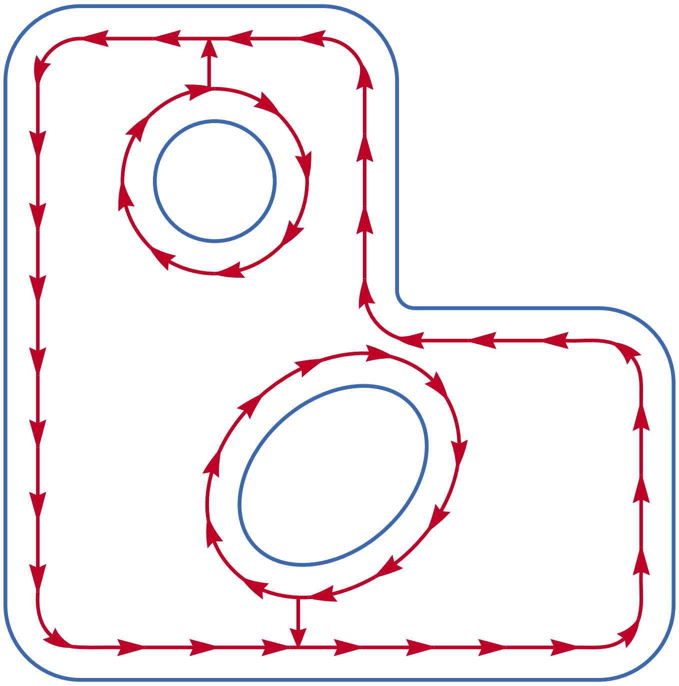
Constant-Z Contour parameters

Exit material

This option controls the tool movements between the working areas.

Connect islands

Choosing the **Connect islands option** enables you to keep the same cutting direction (conventional or climb milling) throughout the entire tool path where possible. This is particularly important in high-speed cutting.



Start from:

- **Inside**

  This option allows you to work in a pocket area starting from the middle of the pocket and cutting towards the outside border of the pocket.

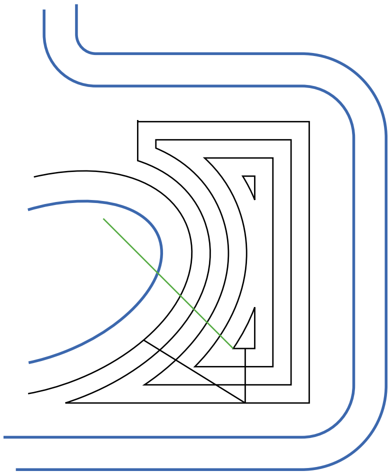
- **Outside**

  This option allows you to work in a pocket area starting from the outside border of the pocket and cutting towards the middle of the pocket.

Corner

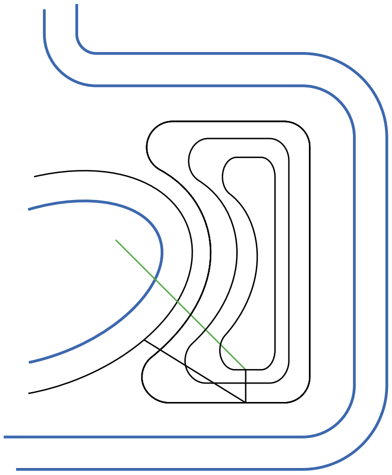
- **None**

The tool path will be formed with sharp corners between one direction and the next.



- **Fillet Corner**

Choosing **Fillet** will connect each direction with a given radius allowing for a smoother transition between path directions.



- **Min corner radius.**

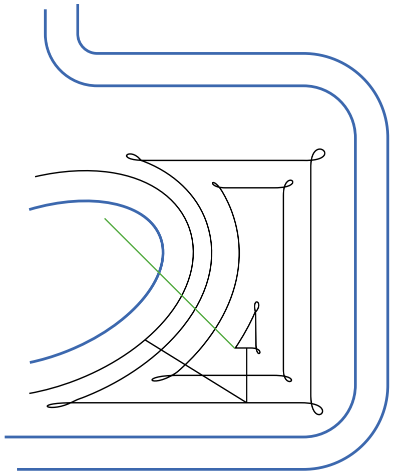
SolidCAM enables you to define the minimal value of the corner radius.

If the given radius is too big for a specific corner, it will produce the biggest possible radius at that point.

There are times were the fillet option might leave some material. This is particularly true if the given radius is large.

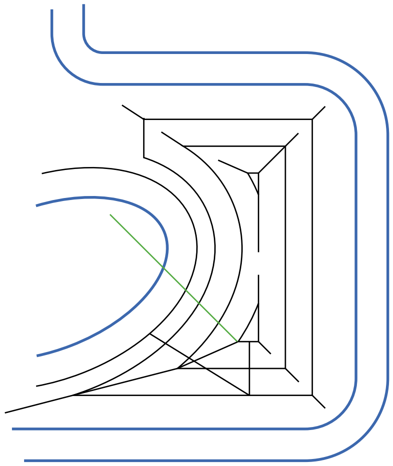
- **Smooth Corner**

The **Smooth Corner** option is another cornering option that ensures that the connecting points on the path will be cleaned using a smooth transition. The tool path will form a loop in the corner, preventing an abrupt change of direction.



- **Sharp Corner**

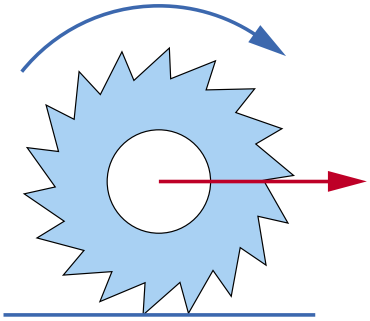
The **Sharp Corner** option ensures that using a short and simple movement will clean the connecting points on the path. Although this produces a sharp movement by the tool, the path itself is slightly shorter than the smooth corner option. This can help cut down on machining time.



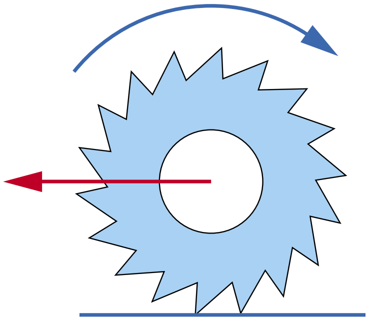
This option is not recommended for high-speed cutting.

Direction

Enables you to choose climb or conventional milling for the roughing operation.



Climb Milling



Conventional Milling

Arc approximation

This command enables you to create G2/G3 G-Code output from Contour rough operations. SolidCAM checks whether successive points of the calculated tool path can be connected using an arc or circle. If an arc or circle connection within the specified arc approximation tolerance can be made, you will receive arc and circle interpolation commands G2 and G3 in the generated G-Code.

This feature can drastically reduce the number of lines in G-Code files. Most CNC-controllers and machines work much faster on arcs and circles than on single tool path points or splines. Arc approximation will increase actual feed rates on older CNC-machines and the machine will work smoother.

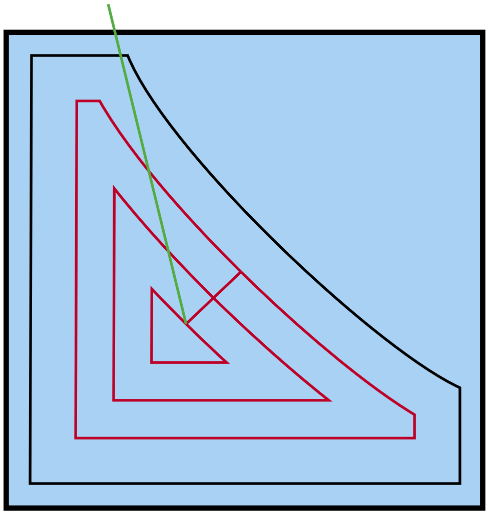
The **Tolerance** value defines the tolerance SolidCAM uses to position tool path points on arcs or circles.

Adjacent tool path connection

This option enables you to choose the method that the tool will move within a pocket from one tool path to the next.

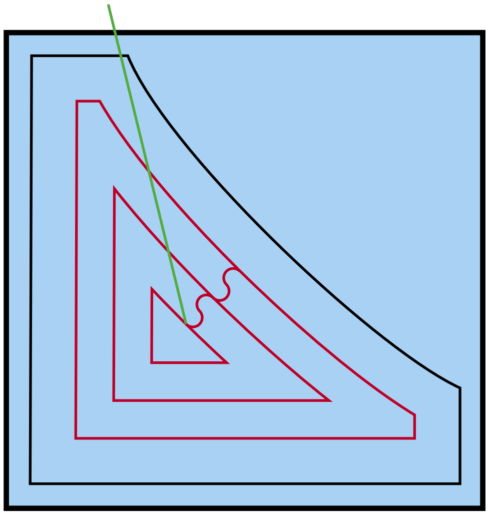
- **Linear connection**

  This will move the tool in a normal approach from one tool path to the next.



- **Rounded connection**

  This will move the tool in an arc path from one tool path to the next. The size of the arc is half the distance between tool paths.



Plunging

This option is relevant only for the **Contour strategy** in the **Pocket Job**.

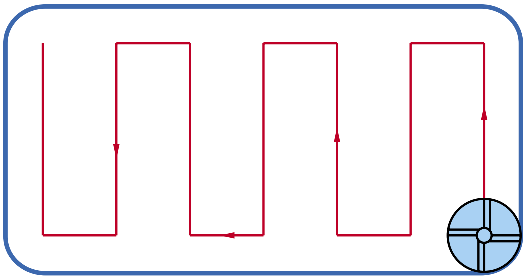
With this option SolidCAM enables you to combine the traditional contour strategy with modern plunging technology. The **Plunging dialog** will be displayed (see the topic 6.2.8.).

Trochoidal milling

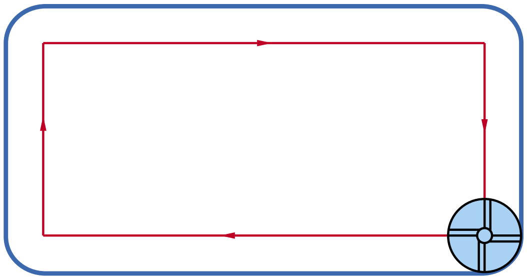
This strategy enables you to change straight tool movements with trochoidal movements. Click on the **Trochoidal** button. The **Trochoidal data dialog** (see the topic **6.1.13**.) will be displayed to specify the necessary parameters.

6.2.6 Hatch+Finish type

With this option, SolidCAM first machines the pocket on the actual down step with the **Hatch** technology and then makes a final path around the pocket contour.

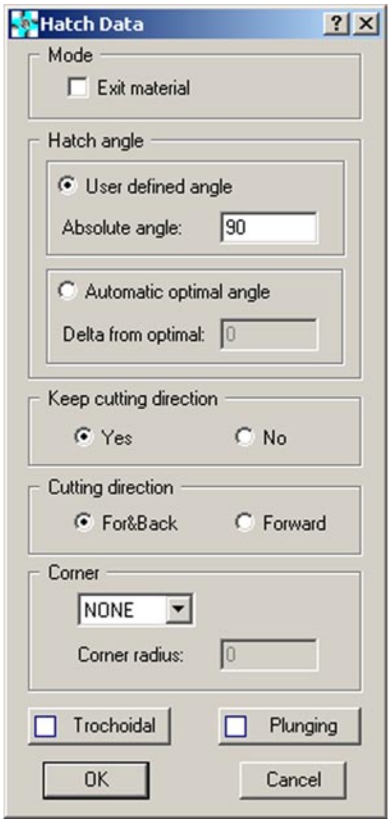


Hatching



Final profiling

Click on the **Data** button. The **Hatch Data dialog** will be displayed.



Hatch Data

Mode
☐ Exit material

Hatch angle
● User defined angle
Absolute angle: 90
○ Automatic optimal angle
Delta from optimal: 0

Keep cutting direction
● Yes ○ No

Cutting direction
● For&Back ○ Forward

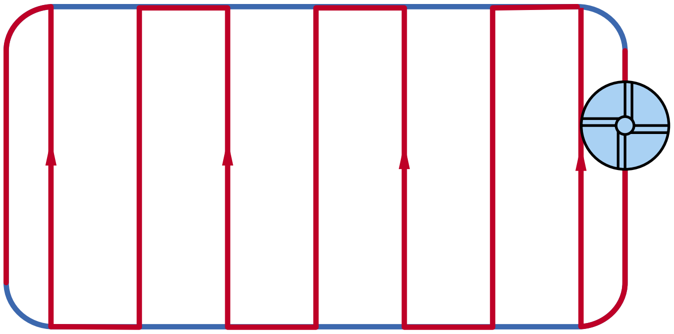
Corner
NONE
Corner radius: 0

For more information on the **Hatch Data dialog**, please refer to the topic **6.2.4**.

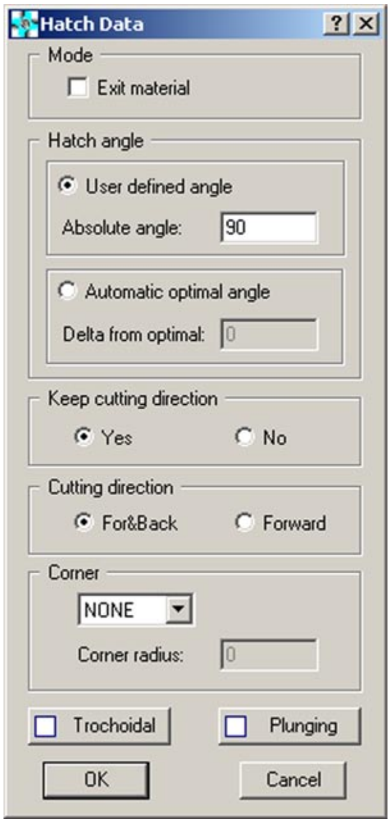
6.2.7 Clear type

This option enables SolidCAM to machine an area that is half a tool diameter larger than the pocket geometry in a hatch-style linear pattern. This enables you to plane the upper face of a workpiece.

Existing pocket islands will be ignored and the whole pocket area will be cleared.



Click on the **Data** button. The **Hatch Data dialog** will be displayed.



Hatch Data

Mode
☐ Exit material

Hatch angle
● User defined angle
Absolute angle: 90
○ Automatic optimal angle
Delta from optimal: 0

Keep cutting direction
● Yes ○ No

Cutting direction
● For&Back ○ Forward

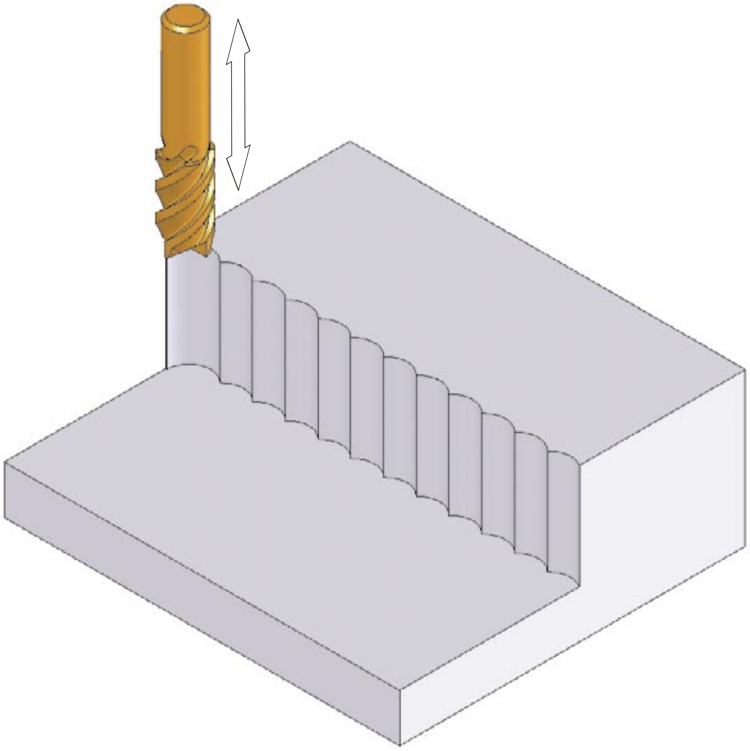
Corner
NONE
Corner radius: 0

☐ Trochoidal ☐ Plunging

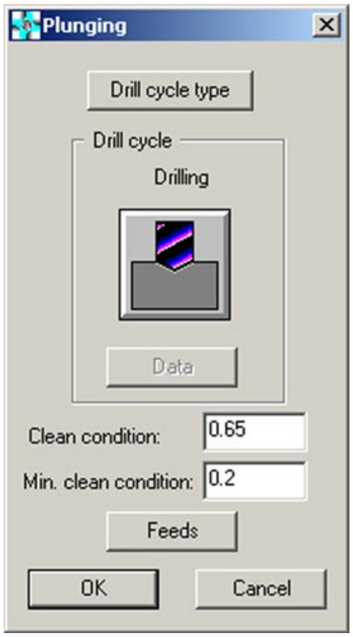
For more information on the **Hatch Data dialog**, please refer to the topic **6.2.4**.

6.2.8 Plunging Pattern

Plunging is a totally different concept of removing material from a given pocket with a special tool. Instead of milling the material, the tool moves up and down as in a drilling motion, traveling along the path type you choose.



Click on the **Data** button. The **Plunging Data dialog** will be displayed.


Plunging
Drill cycle type
Drill cycle
Drilling
Data
Feeds
OK
Cancel

Clean condition: 0.65

Min. clean condition: 0.2

Drill cycle type

SolidCAM enables you to choose preferable drilling cycles for the plunging operation.

1. Click on the **Drill cycle type** button and choose the cycle type you want in the Drill cycle dialog.
2. Click on the **Data button** and specify the parameters in the **Drill options dialog**.

For more information on the **Drill cycles**, please refer to the topic **6.3.4**.

Clean condition

This determines the maximum percentage of material that will be removed based on the diameter of the tool.

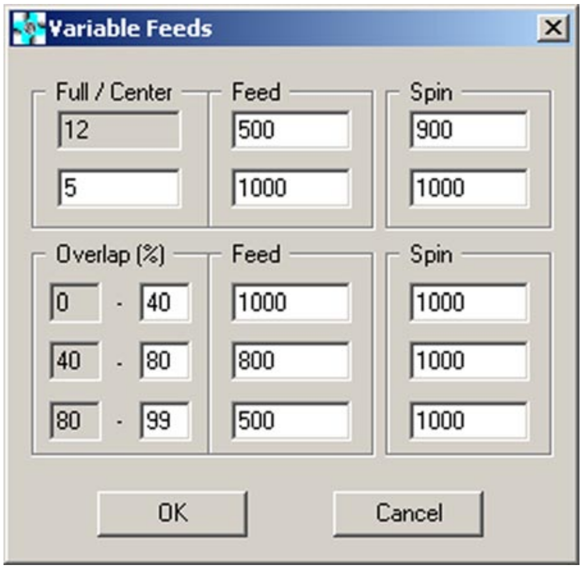
Minimal clean condition

This determines the minimum percentage of material that will be removed based on the diameter of the tool.

Feeds

The plunging feeds can be determined by the percentage of the tool and the diameter of the tool being used.

Full/Center:



Full / Center	Feed	Spin
12	500	900
5	1000	1000

Overlap (%)		Feed	Spin
0	40	1000	1000
40	80	800	1000
80	99	500	1000

If the full diameter of the tool is cutting, the program will use the feed and speed that was placed in the appropriate field.

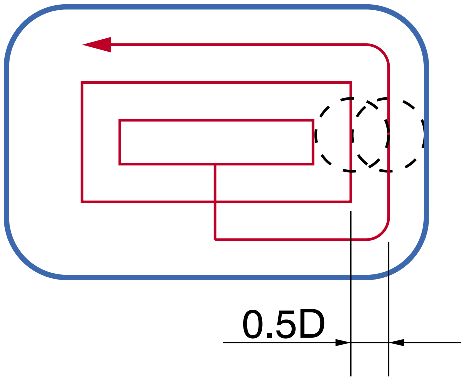
In the second field, you can determine a specific working diameter of the tool. If the working diameter from the center of the tool is greater than this value but less than the full diameter, the feed and spin rate that was entered in the appropriate field will be used.

Overlap (%):

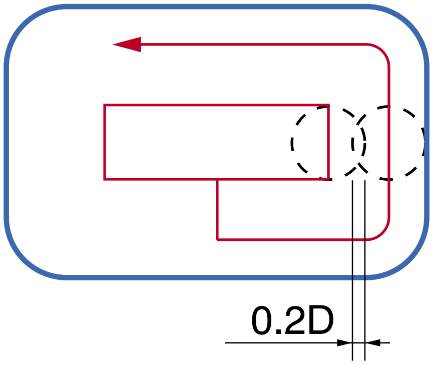
If none of the above conditions have been filled, then the feed and speed will be determined by the overlap of the tool.

6.2.9 Overlap

Enter the **Overlap** of adjacent tool paths in the milling of the pocket. The default value is **0.65** (**65%**). With a tool diameter 10, this would mean a side step of 3.5 for the next tool movement.



Overlap 0.5D



Overlap 0.2D

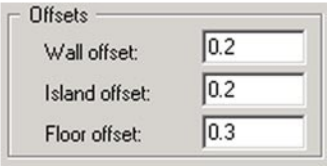
6.2.10 Compensation

If **Compensation** is activated, the tool radius compensation options G4x of the CNC controller are used in the G-Code.

For more information on this subject, please refer to the topic **6.1.9**.

6.2.11 Offsets

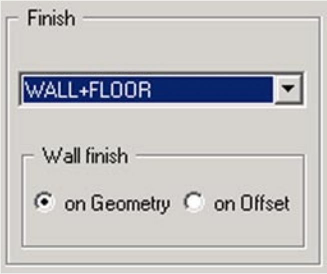
To rough the pocket, you have to specify various offsets on the pocket geometry.



Wall offset: 0.2
Island offset: 0.2
Floor offset: 0.3

- **Wall Offset**. Enter the remaining roughing offset on the wall of the pocket.
- **Island Offset**. Enter the remaining roughing offset on the pocket's island or islands.
- **Floor Offset**. Enter the remaining roughing offset on the floor of the pocket.

6.2.12 Finish



Finish

WALL+FLOOR

Wall finish

● on Geometry ○ on Offset

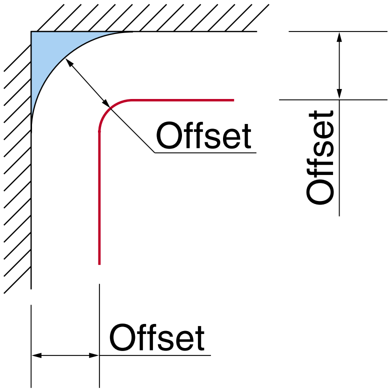
This option is relevant only in case you have specified either a wall, island or floor offset.

- **None**. Use this option when no finishing cut is required. The specified offsets will remain on the pocket contour and floor.
- **Wall**. Only wall and islands will be finished on the contour. The pocket depth will be reduced by the specified floor offset.
- **Floor**. A finish cut is executed to remove the floor offset. Specified Wall and Island offset will remain on the pocket contour.
- **Wall+Floor** A finish cut is executed to remove the wall, island and floor offsets.

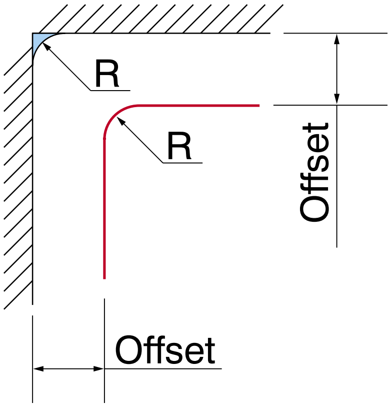
Wall finish

The **Wall finish** field has two options to control the material removal on the last finishing cut.

- **On Offset**. The finish path will be executed at an offset, equal to the sum of the tool radius and the finish offset from the tool path of the last roughing cut. This will result in a uniform cutting depth (side step) on the last cut since in pocket corners the tool will not be drawn into the material.



- **On Geometry**. The last cutting path will be executed exactly on the pocket geometry.

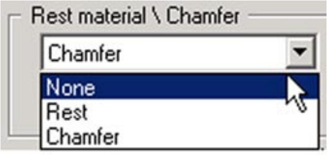


6.2.13 Fillet size for last cut

This option enables you to add a radius to a sharp corner without having to change the geometry.

For more information on this subject, please refer to the topic **6.1.4**

6.2.14 Rest Material/Chamfer



- **Rest material** - This option enables you to define parameters to remove the rest material left unmachined after the previous Jobs.
- **Chamfer** - This option enables you to machine chamfers with the Center drill tool.

For more information on this subject, please refer to the topic **6.1.7**.

6.2.15 Approach

This field controls the way the tool plunges into the material inside the pocket.

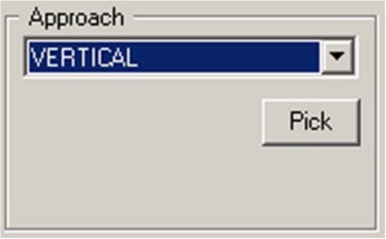
Choose the **Approach** mode and set the approach parameters.

None

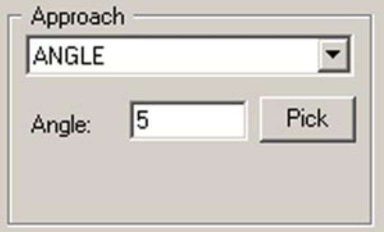
The tool enters the material vertically at the pocket start point automatically chosen by the SolidCAM pocket algorithm.

Vertical

The tool enters the material vertically at a user-defined position. From this position the tool then moves to the pocket's start point, calculated by the pocket algorithm. Use the **Pick** button to specify the position where the tool plunges into the material.



Angle

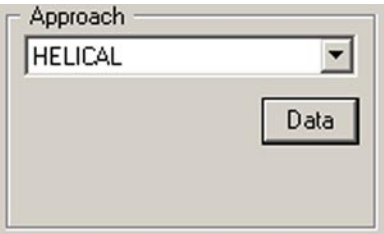

Approach: ANGLE

Angle: 5

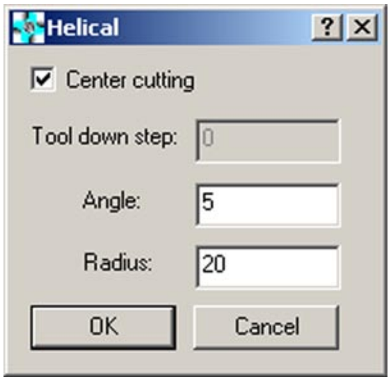
The tool moves to the pocket's start point at a specified ramp angle. The start point must be selected using the **Pick** entry point button. Enter the approach angle into the Angle edit field.

SolidCAM does not check the approach movement against the pocket contour. Check the tool path simulation to make sure that the tool does not gouge the pocket's walls or islands.

Helical


Approach: HELICAL

Click on the **Data** button to set the helical approach parameters.


Helical

☑ Center cutting

Tool down step: 0

Angle: 5

Radius: 20

Tool with Center cutting capabilities

If your cutting tool has Center cutting capabilities, select **Center cutting**. In the **Angle** field, enter the descent angle that you would like the tool to follow. In the **Radius** field enter the radius of the tool path.

The working order will be as follows:

- The **Helical Approach** will descend from the safety distance above the Job upper plane to the material in a circular motion until the down step is reached using the user-defined radius.
- When the tool reaches the down step depth, it will then machine all the material at the down step depth.
- When the machining is completed at this depth, it will then go up to the clearance plane.
- At this stage it will reposition itself at the **Z** safety distance above the previous down step depth and repeat the helical approach to the next working depth.
- This process will repeat itself until the final depth has been machined.

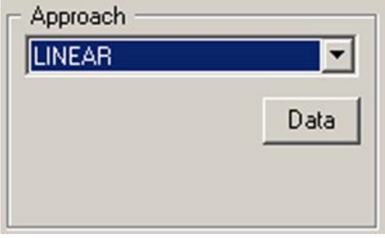
Tool without Center cutting capabilities

If the cutting tool does not have Center cutting capabilities, leave the **Center cutting** field empty. In the **Tool down step** field, enter the depth of the down step of the tool.

The result of this action is the same as with a center cutting tool except for the following differences:

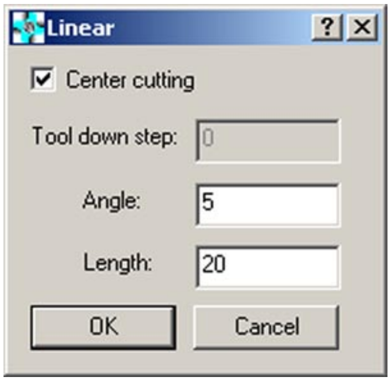
- The tool descends from the safety distance above the Job upper plane to the material in a circular motion up to the **Tool down step**.
- At this point the tool will clear a circular path. It will then descend to the next **Tool down step**.
- This will be repeated until the tool reaches the CAM-Parts down step depth.
- When the tool reaches the down step depth, it will then machine all the material on that down step depth.
- When the machining is completed at this depth, it will then go up to the clearance plane.
- At this stage it will reposition itself at the **Z** safety distance above the previous down step depth and repeat the helical approach to the next working depth.
- This process will repeat itself until the final depth has been machined.

Linear



Approach: LINEAR

The **Linear** approach follows the same rules as the **Helical** approach. The difference is that the descent will be in a linear zigzag fashion rather than a circular one.



☑ Center cutting

Tool down step: 0

Angle: 5

Length: 20

Tool with Center cutting capabilities

If your cutting tool has Center cutting capabilities, select **Center cutting**. In the **Linear** dialog window enter the length of the linear tool path that you would like the tool to travel.

The working order will be as follows:

- The **Linear Approach** will descend from the safety distance above the Job upper plane to the material in a zigzag motion based on the user defined angle and length until the down step is reached.
- When the tool reaches the down step depth, it will then machine all the material on that down step depth.
- When the machining is completed at this depth, it will then go up to the clearance plane.
- At this stage it will reposition itself at the Z safety distance above the previous down step depth and repeat the helical approach to the next working depth.
- This process will repeat itself until the final depth has been machined.

Tool without Center cutting capabilities

If the cutting tool does not have Center cutting capabilities, leave the **Center cutting** field empty. In the **Tool down step** field, enter the depth of the down step you want the tool go to.

The result of this action is the same as with a center cutting tool except for the following differences:

- The tool descends from the safety distance above the Job upper plane to the material in a zigzag motion up to the T**ool down step**.
- At this point the tool will clear a straight linear path. It will then descend to the next **Tool down step**.
- This will be repeated until the tool reaches the CAM-Part down step depth.
- When the tool reaches the down step depth, it will then machine all the material on that down step depth.
- When the machining is completed at this depth, it will then go up to the clearance plane.
- At this stage it will reposition itself at the **Z** safety distance above the previous down step depth and repeat the helical approach to the next working depth.
- This process will repeat itself until the final depth has been machined.

6.2.16 Retreat

This field controls the movement of the tool when it leaves the pocket contour on the finishing cut.

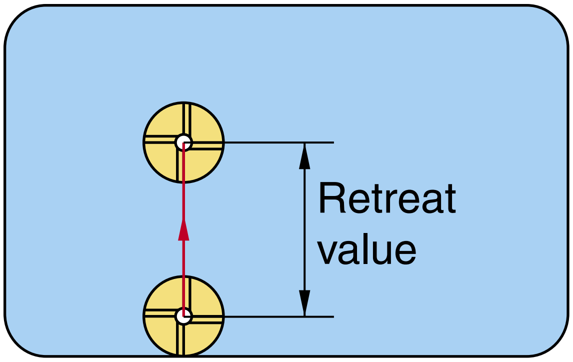
None

On the last point of the contour, the tool exits vertically in rapid move.

This option causes the tool to exit while in contact with the pocket wall, which might cause gouging and could result in visible marks on the workpiece.

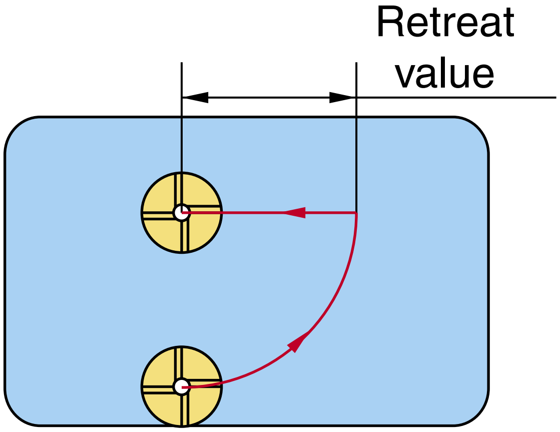
Normal

With a movement normal to the pocket contour, the tool retreats from the last point of the profile. The length of the normal must be specified using the **Distance** value. The tool then rapids vertically to the Job Clearance plane.

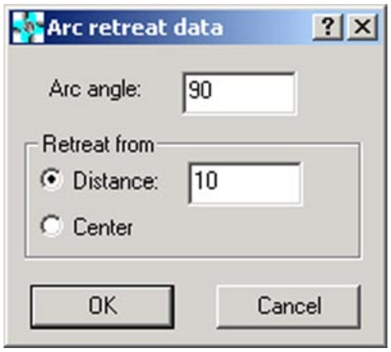


Arc

With a circular motion, tangent to the last entity of the pocket contour, the tool retreats from the profile. The radius of the arc must be specified using the **Distance** value. The tool then rapids vertically to the Job Clearance plane.



The **Data** button will be displayed. Click on the button to display the **Arc retreat data dialog**.



Arc angle: 90

Retreat from

● Distance: 10

○ Center

Arc angle

This parameter defines the angle of the retreat arc segment. The default angle value is 90°; in this case SolidCAM generates an retreat path of quarter arc.

Retreat from

With the **Arc retreat** option the tool moves normal to the retreat arc at the arc start-point.

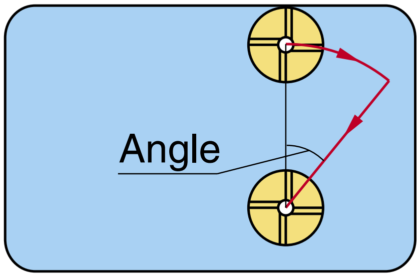
The following options are available:

Distance

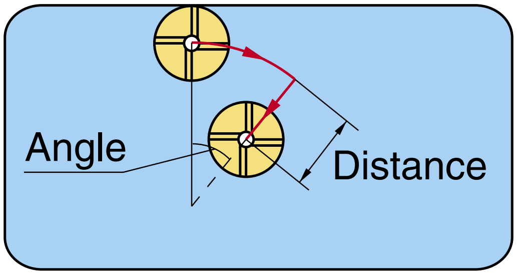
In this case the tool starts from the specified distance from the arc start-point.

Center

In this case the tool starts from the retreat arc center.



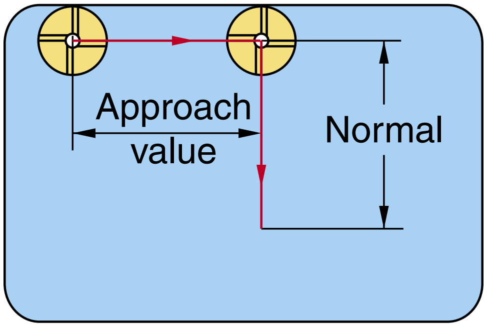
Retreat from **Center**



Retreat from **Distance**

Tangent

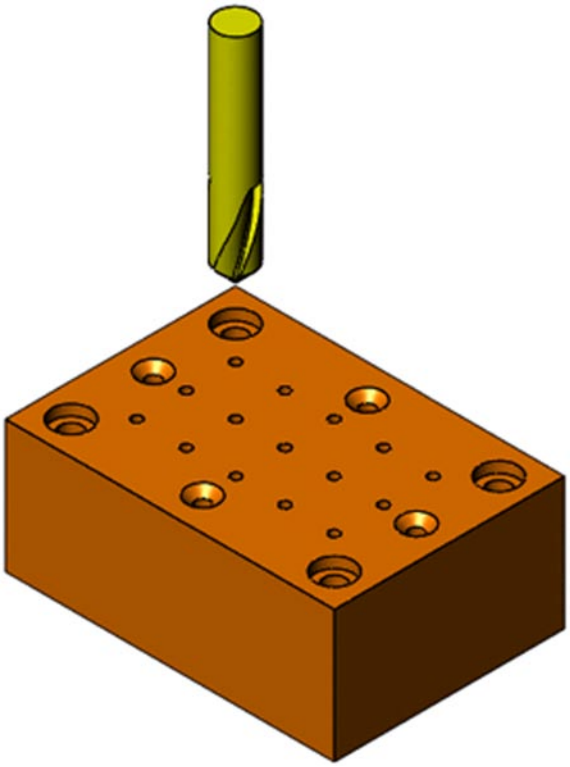
The tool retreats from the pocket wall in a movement tangent to the last profile entity. The length of the tangent must be entered in the **Distance** field.



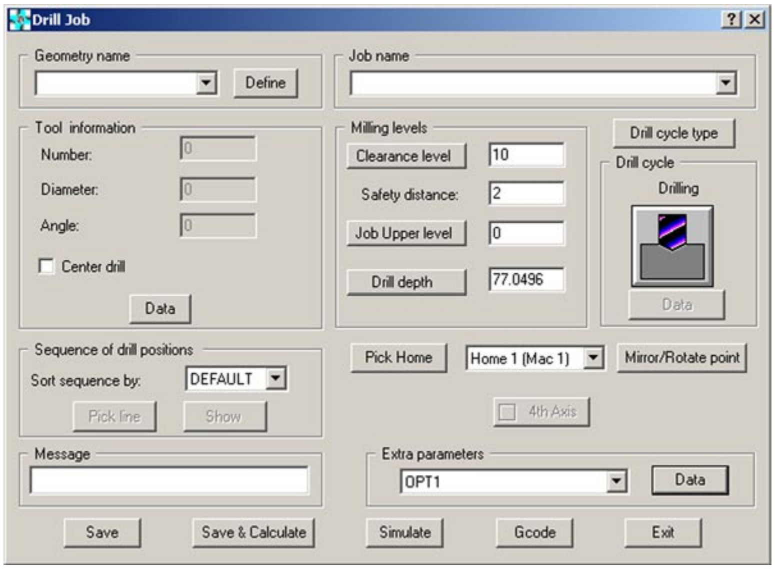
SolidCAM does not check the retreat movements against the pocket contour. Check the tool path simulation to make sure that the tool does not gouge the pocket‘s walls or islands.

6.3 Drill Job

This job enables you to perform **drills** and other canned drill cycles. SolidCAM supports the canned drill cycles provided by your particular **CNC Machine** such as threading, peck, ream, boring etc... If your **CNC Machine** has no canned drill cycles of its own, they can be defined using the **General Pre** and **Post-processor** program (**Gpptool**).



When you select the commands **Add>Drill**, the **Drill Job** screen will be displayed.



Drill Job
Geometry name
Define
Job name
Tool information
Data
Sequence of drill positions
Pick line
Show
Message
Milling levels
Drill cycle type
Data
Pick Home
Mirror/Rotate point
Data
Save
Save & Calculate
Simulate
Gcode
Exit

Number: 0
Diameter: 0
Angle: 0
☐ Center drill
Sort sequence by: DEFAULT
Clearance level: 10
Safety distance: 2
Job Upper level: 0
Drill depth: 77.0496
Drill cycle: Drilling
Home 1 (Mac 1)
☐ 4th Axis
Extra parameters: OPT1

6.3.1 Geometry name

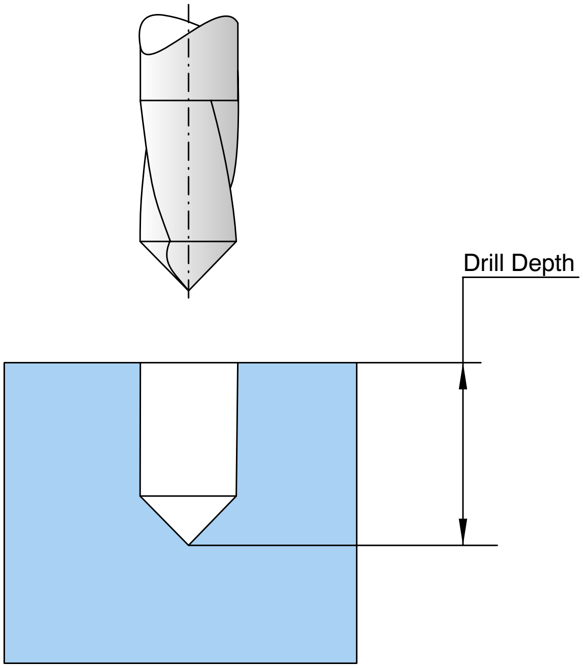
In this section, specify the drill positions, i.e. drill geometry you want to use.

6.3.2 Tool

In addition to the standard tool options described earlier, SolidCAM enables you to label a tool as a center drill. This option is used for automatic sorting. In automatic sorting, tools that are labeled as center drills can be automatically moved to the top of the job list.

6.3.3 Drill depth

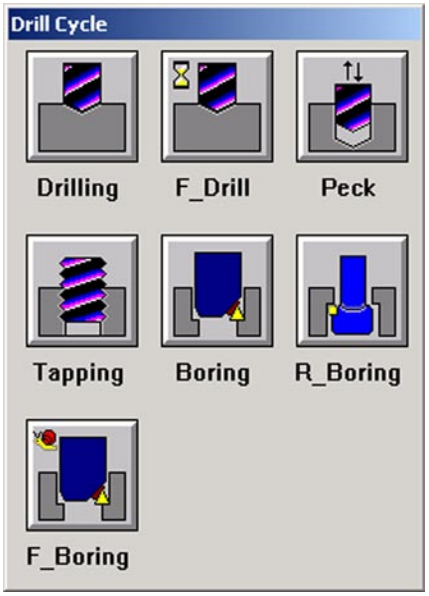
This parameter defines the final depth of the drill.

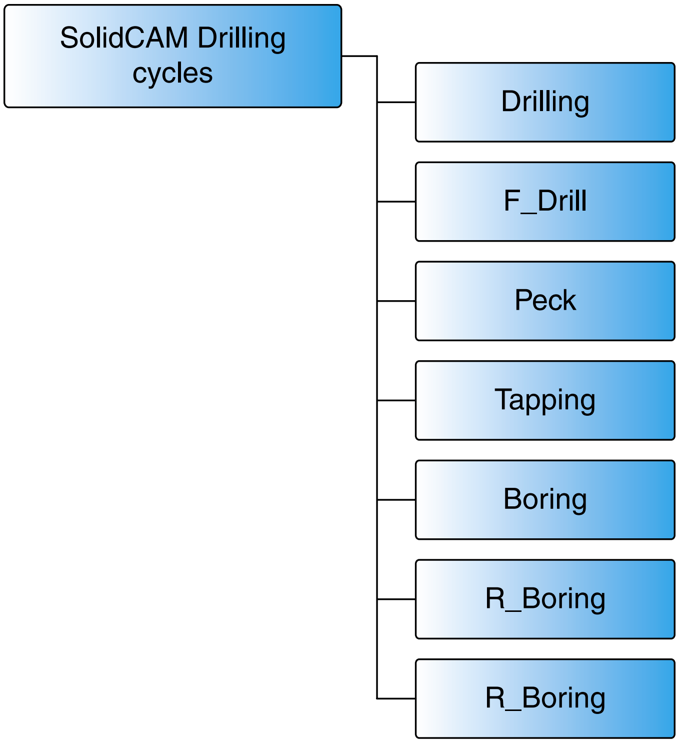


Drill Depth

6.3.4 Drill cycle type

Click on the **Drill cycle** type button. The **Drill canned cycles** supported by the post-processor of the current CNC-controller will be displayed. Click on one of the icons to select the canned cycle you want to use in this job.


Drill Cycle
Drilling
F_Drill
Peck
Tapping
Boring
R_Boring
F_Boring


SolidCAM Drilling cycles
Drilling
F_Drill
Peck
Tapping
Boring
R_Boring
R_Boring

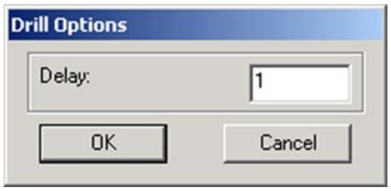
Drill canned cycles must be defined in the *.mac and *.gpp post-processor of the currently active CNC-controller in your CAM-Part. Contact customer support to learn more about customizing your post-processor.

Drilling

With this option the drill will travel in one single motion to the specified depth and retract.

F_Drill

With this option the drill will travel in one single motion to the specified depth, pause for a specified amount of time and then retract.



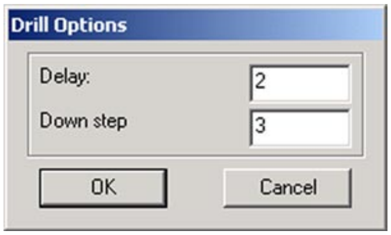
Delay: 1

- **Delay**

  In the **Delay** field, enter the amount of time (in seconds) you want the tool to pause at the depth point of the hole.

Peck

When you choose the **Peck Data** button, the **Peck data dialog** box is displayed. You are prompted to enter the **Down step**, **Release distance** and to choose the **Release type**.



Delay: 2

Down step: 3

- **Down Step**

  This field determines the value of each down step of the drill.

- **Delay**

  In the **Delay** field, enter the amount of time (in seconds) you want the tool to pause at the depth point of the hole.

Tapping

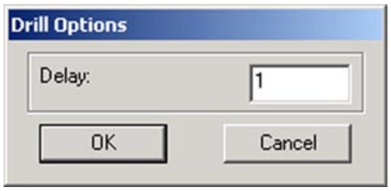
With this option the tap will travel in one single motion to the specified depth and then retract reversing its spin direction.

Boring

With this option the boring tool will travel in one single motion to the specified depth, stop its spin motion and retract rapidly.

R_Boring

With this option the boring tool will travel in one single motion to the specified depth, stop its spin motion, pause for a specified amount of time and retract rapidly.



Delay: 1

- **Delay**

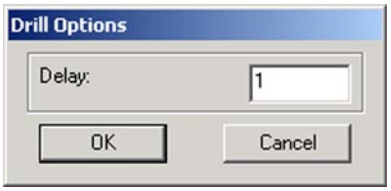
  In the **Delay** field, enter the amount of time (in seconds) you want the tool to pause at the depth of the hole.

F_Boring

With this option the boring tool will travel in one single motion to the specified depth and retract at the same feed rate it entered.

- **Delay**

  In the **Delay** field, enter the amount of time (in seconds) you want the tool to pause at the depth of the hole.



Delay: 1

6.3.5 Sequence of drill positions

You can select one of 3 available options to control the sequence of the drill positions.

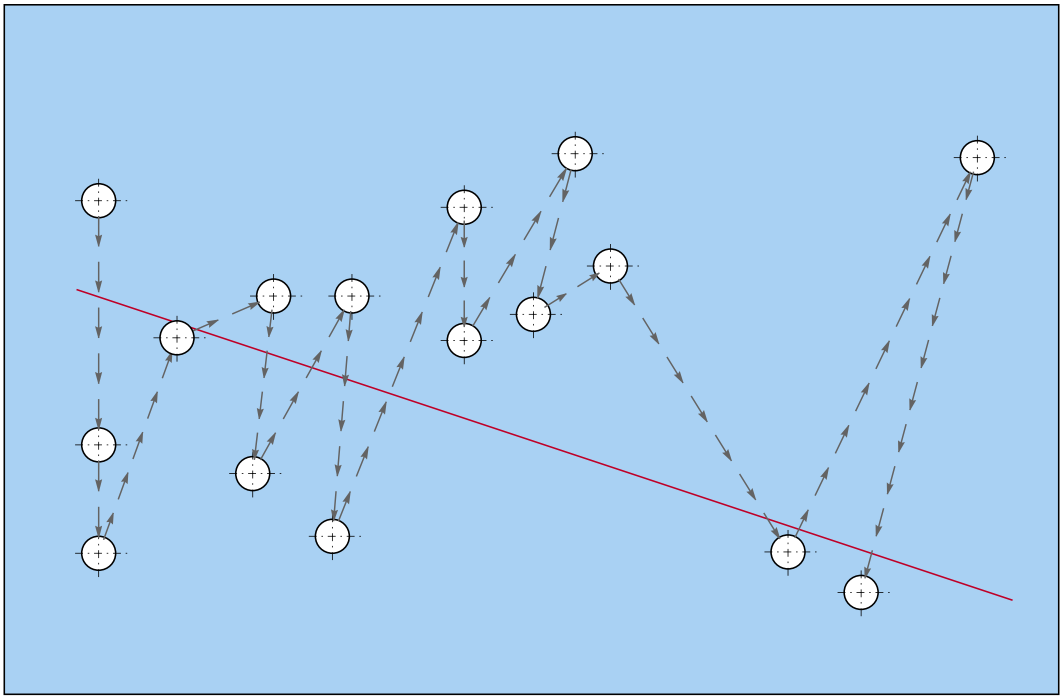
- **Default**

  The drills will be performed according to the order in which the drill points were picked during the geometry definition.

- **Line**

  You have to define a direction by selecting two points on your model.

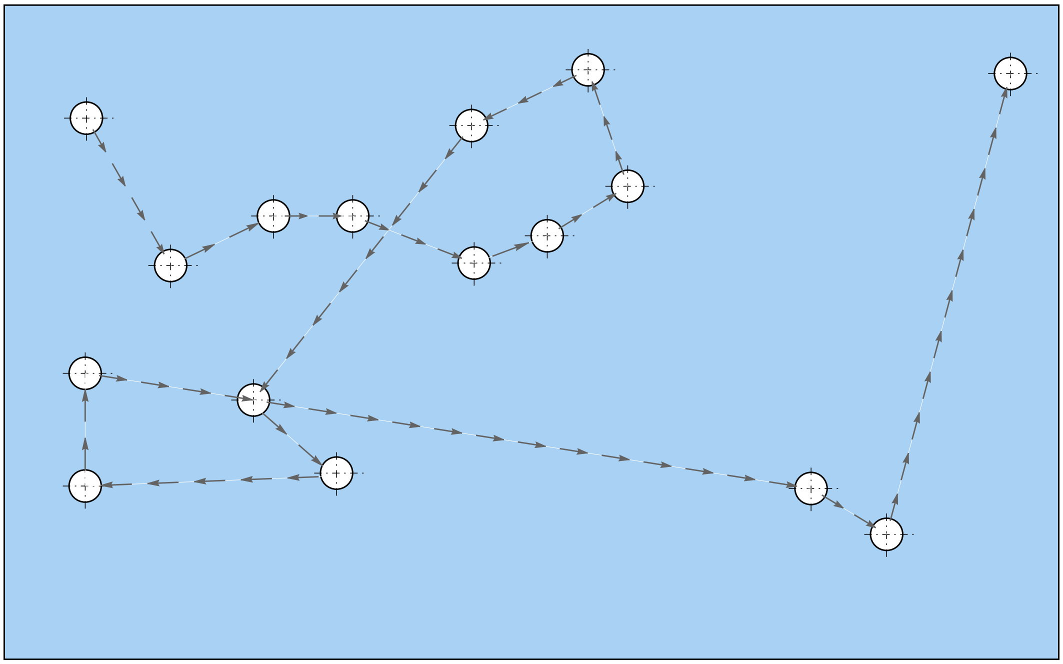
  The drill positions will then be arranged along this **Direction line**.



1. Click on the **Pick line** button.
2. **Pick first point**: Enter the X,Y,Z coordinates into the Edit bar and confirm with Enter or pick a position using the CAD point selection options.
3. **Pick second point**: Select the end point of the direction line.
4. Use the show button to evaluate the result of the sort by line operation.

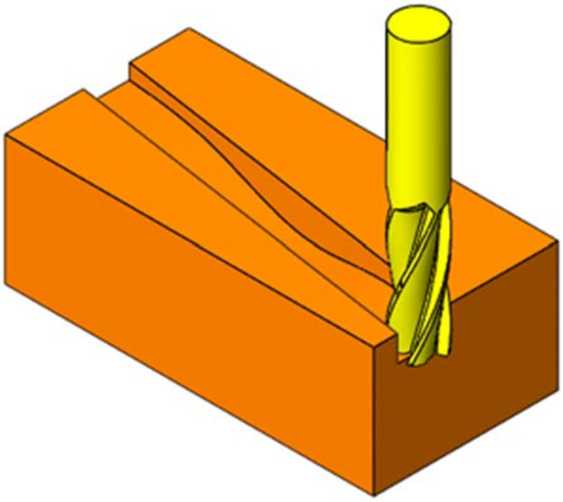
- **Distance**

This option minimizes the length of the necessary tool movement. The first drill will be performed on the first point defined in the drill geometry. Any other drill will be executed on the nearest position, i.e. within the shortest **distance** to the previous drill position.

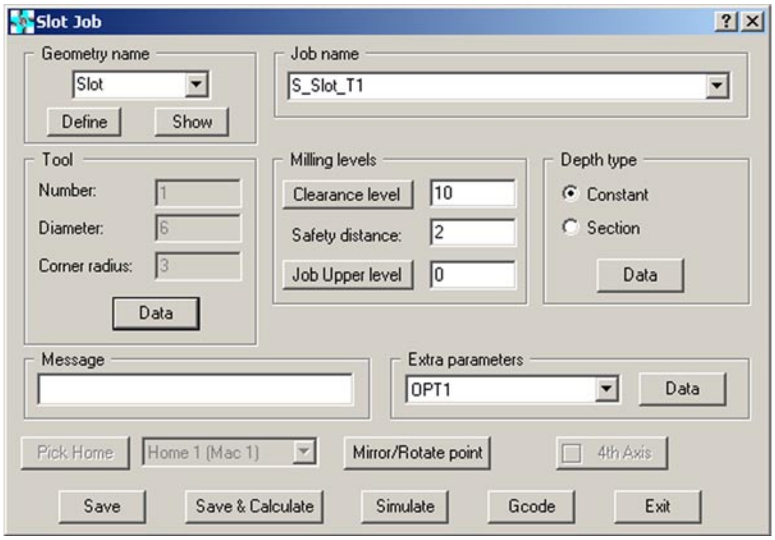


6.4 Slot Job

This job generates a tool path along the centerline to the right or to the left of one or more profiles. Two types of slots can be defined: The job **Slot with constant depth** machines the slot in several down steps until it reaches the final depth. In **Slot with variable depth**, the depth profile is also defined by a 2D section. The slot can be pre-machined using rough and semi-finish cycles. The finish cut will produce a tool path according to the specified scallop height on the floor of the slot. With available parameters for a right and left extension and side step, you can mill a **slot** wider than the tool diameter.



When you select the commands **Add>Slot**, the **Slot Job** screen will be displayed.



Slot Job

Geometry name: Slot

Job name: S_Slot_T1

Tool
Number: 1
Diameter: 6
Corner radius: 3

Milling levels
Clearance level: 10
Safety distance: 2
Job Upper level: 0

Depth type
● Constant
○ Section

Message:

Extra parameters: OPT1

Home 1 (Mac 1)

☐ 4th Axis

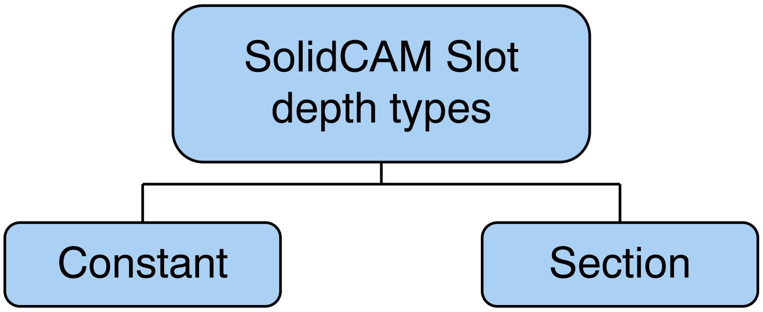
6.4.1 Geometry name

In this section, specify the contour, i.e. profile geometry you want to mill.

6.4.2 Depth type

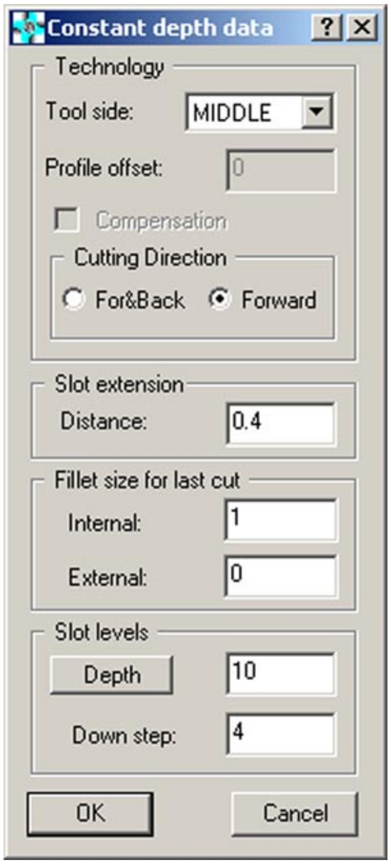
There are two methods to define the depth of the slot.

Select the depth type **Constant** or **Section** and click on the **Data** button to enter additional parameters.



6.4.3 Constant Depth type

This option will produce a slot with a uniform depth. The **Constant Data dialog** will be displayed.


Constant depth data

Technology

Tool side: MIDDLE

Profile offset: 0

☐ Compensation

Cutting Direction

○ For&Back ● Forward

Slot extension

Distance: 0.4

Fillet size for last cut

Internal: 1

External: 0

Slot levels

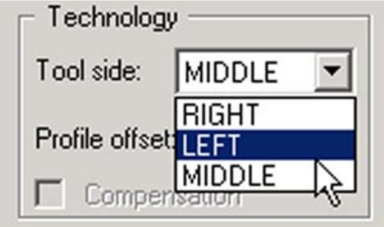
Depth: 10

Down step: 4

Technology parameters

Tool side

Specify on which side of the slot profile you want to mill. The tool side is always relative to the direction of the slot's chain geometry.



Technology

Tool side: MIDDLE

Profile offset:

☐ Compensation

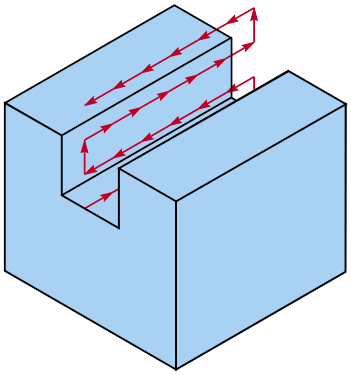
- **Right**. The tool will cut on the right side of the profile geometry at a distance defined by the value in the **Profile Offset**.
- **Left**. The tool will cut on the left side of the profile geometry at a distance defined by the value in the **Profile Offset**.
- **Middle**. The center of the tool will move on the profile geometry. (The cutter radius compensation G4x cannot be used with this option).

The **Profile offset** will remain on the left/right side of the profile after machining.

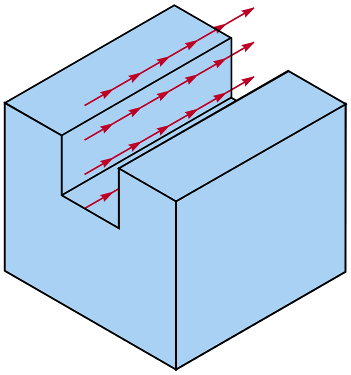
If **Compensation** is activated, the cutter radius compensation options G4x of the CNC controller are used in the G-Code.

Cutting direction

- **Forward** – this option enables you to create the tool path containing only one-directional movements.
- **For&Back** – this option enables you to create the tool path containing bi-directional movements.



For&Back option



Forward option

Slot extension

This option enables you to extend the end of the slot without additional geometric construction and without editing the slot geometry.

- **Distance**. This parameter controls the slot extension.

Fillet size for last cut

This option enables you to add a radius to a sharp corner without having to change the geometry.

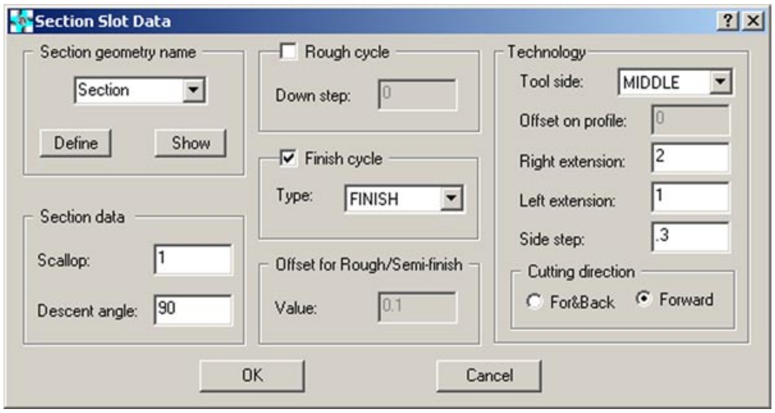
For more information on this subject, please refer to the topic **6.1.4**.

Slot levels

- **Depth**. Enter the depth of the slot relative to the job upper plane. You can use the Depth button to pick the Z-level from the 3D model.
- **Down step**. Enter the cutting depth for two successive cuts.

6.4.4 Section depth type

When you click on the **Data** button, the **Section Slot Data** dialog will be displayed.



Section Slot Data

Section geometry name: Section

Section data
Scallop: 1
Descent angle: 90

☐ Rough cycle
Down step: 0

☑ Finish cycle
Type: FINISH

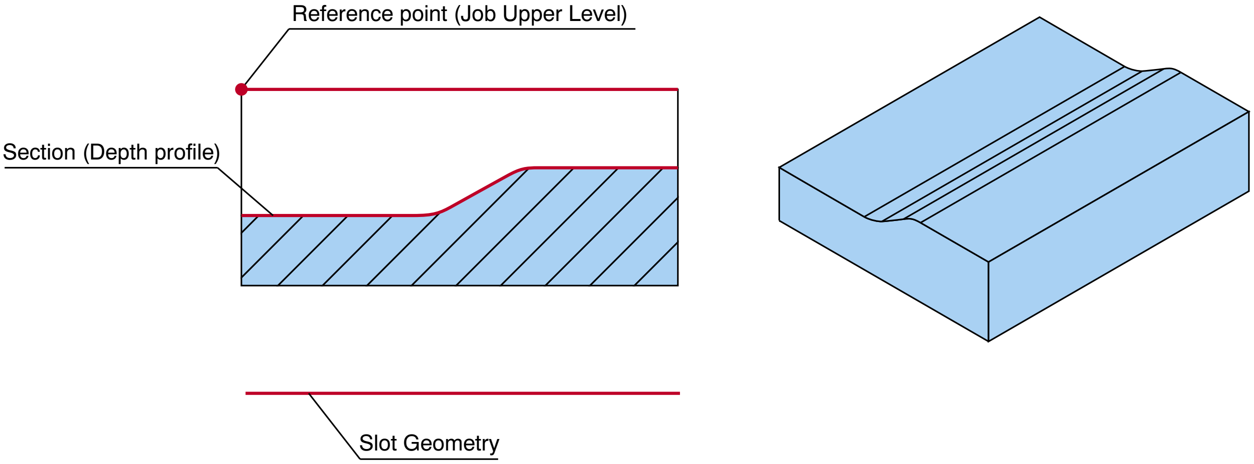
Offset for Rough/Semi-finish
Value: 0.1

Technology
Tool side: MIDDLE
Offset on profile: 0
Right extension: 2
Left extension: 1
Side step: .3

Cutting direction
○ For&Back ● Forward

Section geometry name

The section specifies the depth profile for this slot job. As the geometry for this 2.5D job is designed using **2D profiles and sections**, a reference point is required to indicate the position of the upper job level.



Section data

- **Scallop**

  This value determines the quality of the resulting surface on the floor of the slot. The smaller the value, the smoother the surface will be.

- **Descent Angle**

  The tool will plunge into the material with the given ramp angle. The default value of 90 degrees causes a normal approach into the material. The tool enters the material at an angle to the first slot profile. If the tool path length required to perform the ramp angle is longer than the slot profile, SolidCAM will generate a zig-zag movement to plunge into the material.

Rough, Semi-finish and Finish cycles

- **Rough cycle**

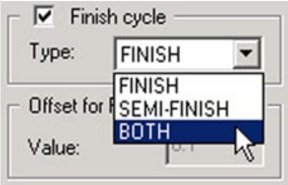
  If you check this option, a rough operation will clear the slot before the semi-finish operation is performed.

  Enter the **Down step** between two successive cuts during roughing.

  Define the remaining offset that will remain on the slot walls and floor after the roughing cut in the field **Rough/Semi-finish offset**.

- **Finish cycle**

Check this option if you intend to semi-finish and/or finish the slot. Use the **List field** to select the cycles you want to perform.



Type: FINISH

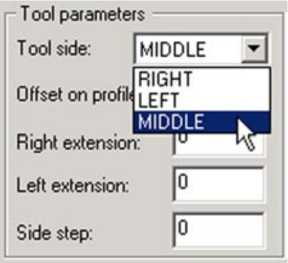
Value:

- **Semi-finish** will prepare the slot at an offset specified in the field **Rough/Semi-finish offset**.
- **Finish** will machine the slot to its final dimensions and specified surface scallop.
- **Both**. Use this option to machine the slot with a semi-finish first and then a finishing cut.

Offset for Rough/Semi-finish. This value specifies the offset that remains on the slot after the rough and semi-finish operations. This offset will be removed in the last finish cut.

Tool Parameters

Use the **List field** to specify on what side of the slot profile you want to mill. The tool side is always relative to the direction of the slot's chain geometry.



Tool side: MIDDLE

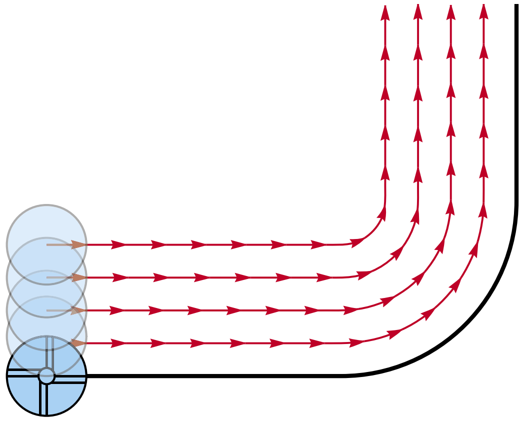
Right extension: 0

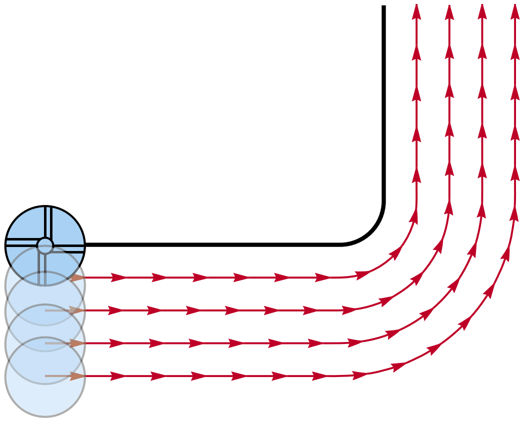
Left extension: 0

Side step: 0

- **Right**. The tool will machine on the right side of the slot geometry. The cut will leave material on the profile equal to the specified Offset in the profile value.
- **Left**. The tool will machine on the left side of the slot geometry. The cut will leave material on the profile equal to the specified Offset in the profile value.
- **Middle**. The center of the tool will move on the profile geometry.

With additional parameters for the **right and left extension,** the Middle option enables you to machine slots that are wider than the tool diameter. The tool will use the given side step value to cut from the right to the left side. Note that the side definition is always relative to the direction of the slot profile. Check the direction of the profile with the Geometry>Show option in the slot job screen.

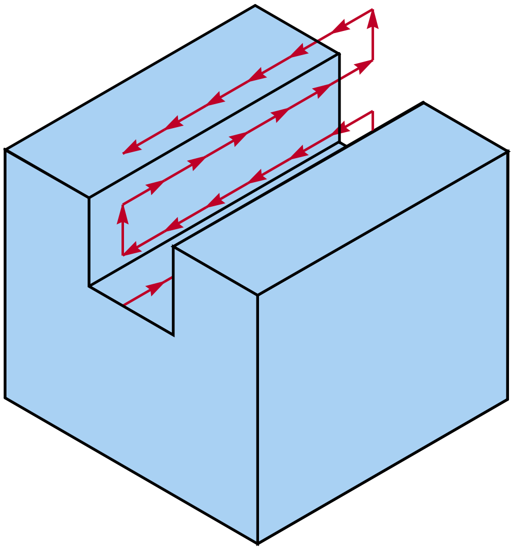




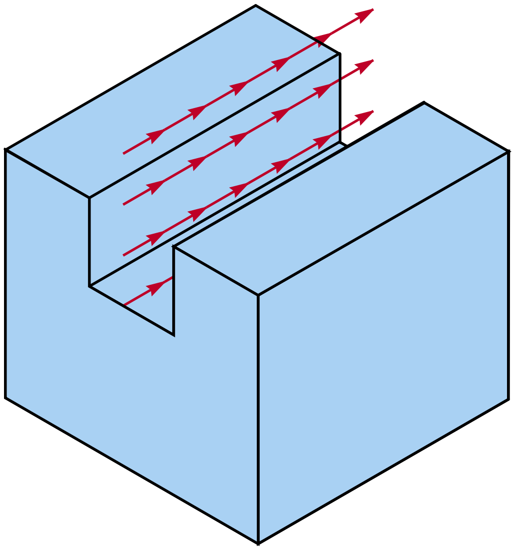
- **Right extension**. Enter the distance by which the slot will be extended to the right side (relative to profile direction).
- **Left extension**. Enter the distance by which the slot will be extended to the left side (relative to profile direction).
- **Side step**. Enter the tool's side step in successive tool paths.

Cutting direction

- **Forward** – this option enables you to create the tool path containing only one-directional movements.
- **For&Back** – this option enables you to create the tool path containing bi-directional movements.



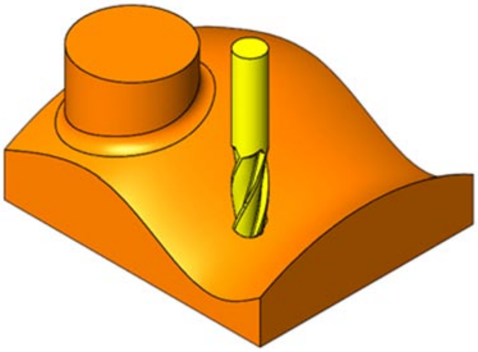
For&Back option



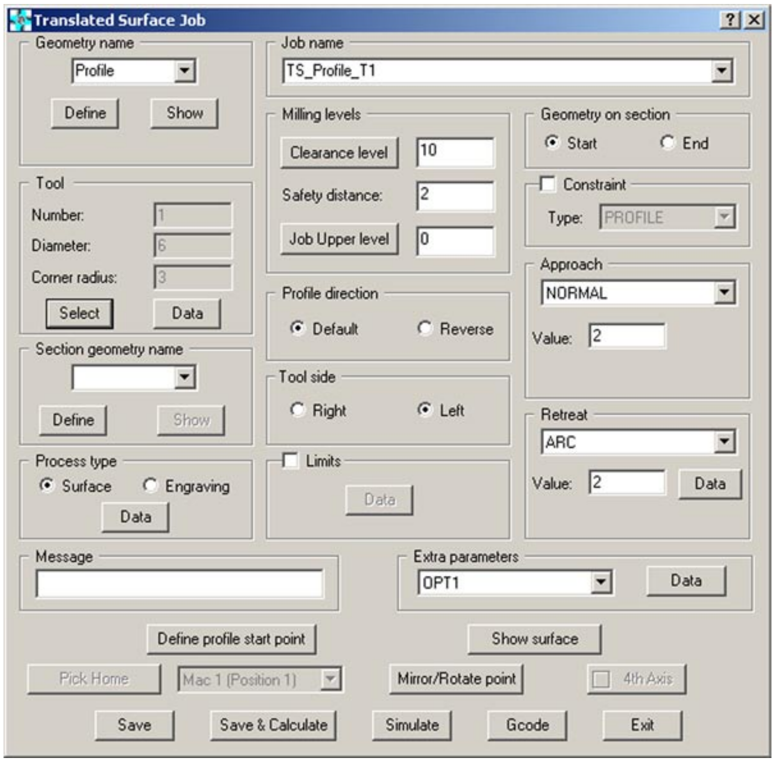
Forward option

6.5 Translated Surface Job

A translated surface is generated by moving a section along a profile geometry. Limit geometries can be projected on the translated surface. You can machine the resulting **translated surface** inside, outside or along the limit geometry.



When you select the commands **Add>Translated Surface,** the **Translated Surface Job** screen will be displayed.


Translated Surface Job
Define
Show
Select
Data
Define
Show
Data
Clearance level
Job Upper level
Data
Data
Data
Define profile start point
Show surface
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

Geometry name: Profile

Job name: TS_Profile_T1

Milling levels
Clearance level: 10
Safety distance: 2
Job Upper level: 0

Geometry on section: ● Start ○ End

☐ Constraint
Type: PROFILE

Tool
Number: 1
Diameter: 6
Corner radius: 3

Approach: NORMAL
Value: 2

Profile direction: ● Default ○ Reverse

Section geometry name:

Tool side: ○ Right ● Left

Retreat: ARC
Value: 2

Process type: ● Surface ○ Engraving

☐ Limits

Message:

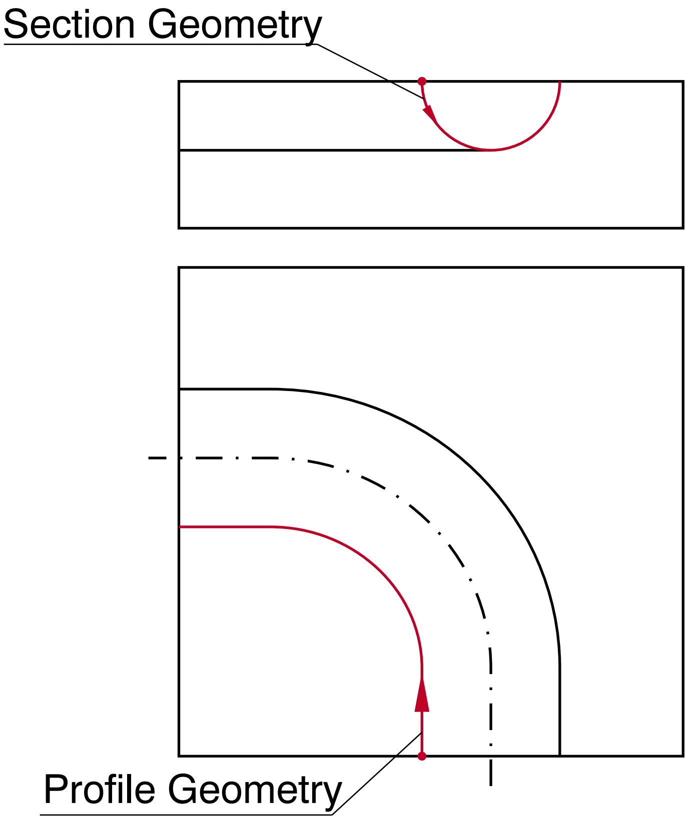
Extra parameters: OPT1

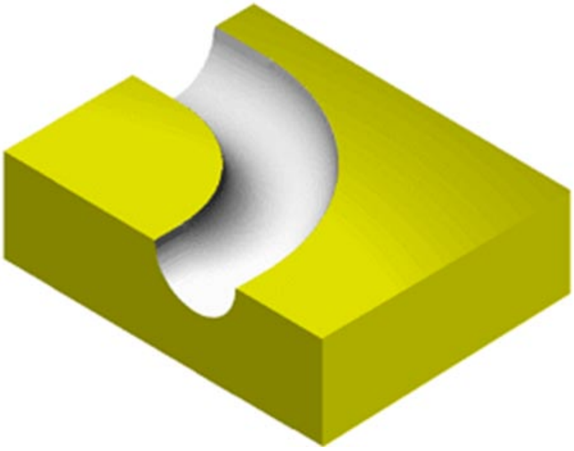
Mac 1 (Position 1)

☐ 4th Axis

6.5.1 Geometry name

A translated surface requires two geometries, the profile along which the section will be translated and the section geometry.


Section Geometry
Profile Geometry



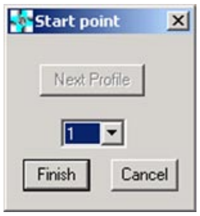
The geometries in the above example will result in this **Translated surface**.

6.5.2 Define Profile Start Point

Define profile start point

This button enables you to pick the start point of the machining in case you use a closed profile geometry only.

1. Click on the **Define profile start point** button. The **Start point dialog** will be shown.



2. Click on the start point on the closed profile.

3. Click on the **Finish** button to confirm the data.

6.5.3 Profile direction

This option controls the direction of the profile geometry for this particular job.

- **Default** will machine the profile in the direction the chain has been defined.

- **Reverse** changes the direction of the geometry for this job.

6.5.4 Tool side

The tool side determines on which side of the profile the section will be machined. Use the **Show surface** button to display the current tool position relative to the profile geometry.

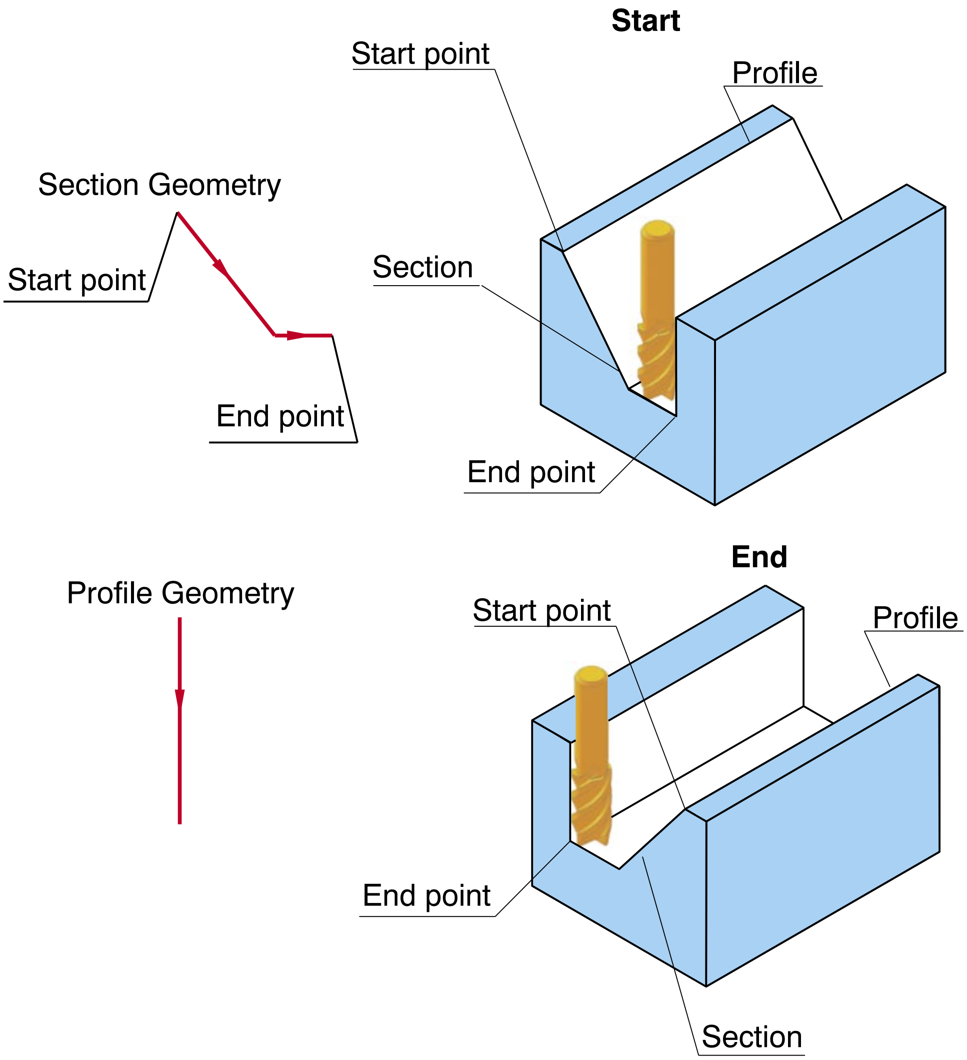
The geometry direction set in the **Reverse geometry** field also affects the tool side option.

- **Right**. The tool path is generated on the right hand side of the profile geometry.

- **Left.** The tool path is generated on the left hand side of the profile geometry.

6.5.5 Geometry on Section

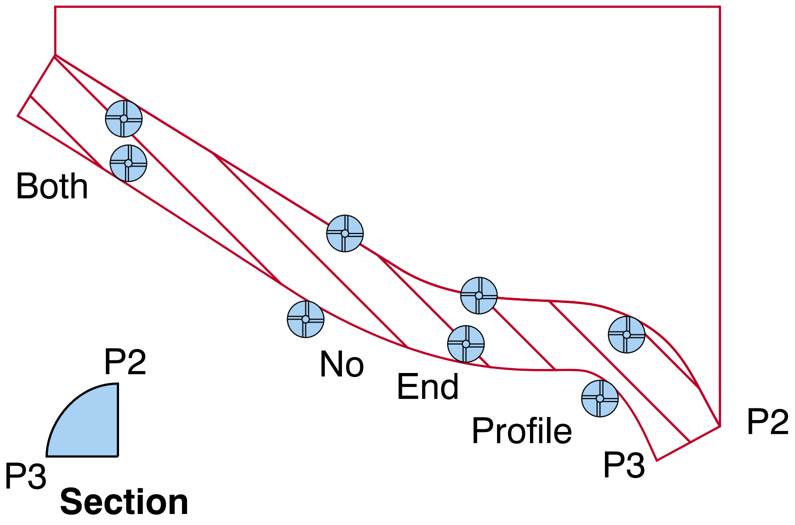
The **Geometry on Section** option enables you to locate the section on the profile.

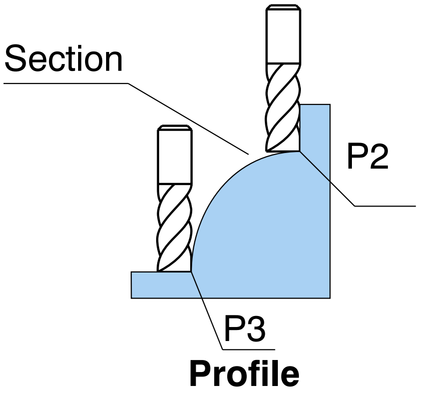

Start
Start point
Profile
Section Geometry
Start point
Section
End point
End point
End
Profile Geometry
Start point
Profile
End point
Section

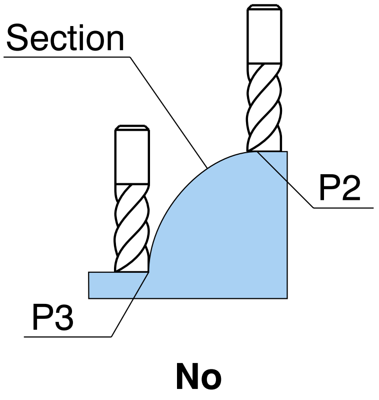
- **Start** will locate the profile geometry at the start of the section and the machining will be done from the start of the section towards the end of the section.
- **End** will locate the profile geometry at the end of the section and the machining will be done from the end of the section towards the start of the section.

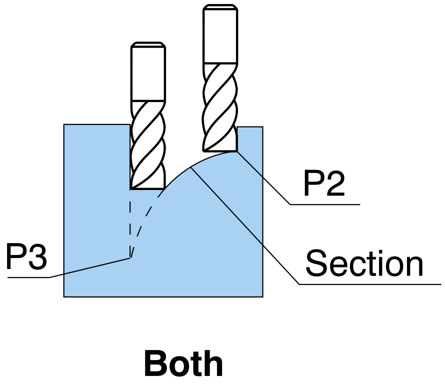
6.5.6 Constraint

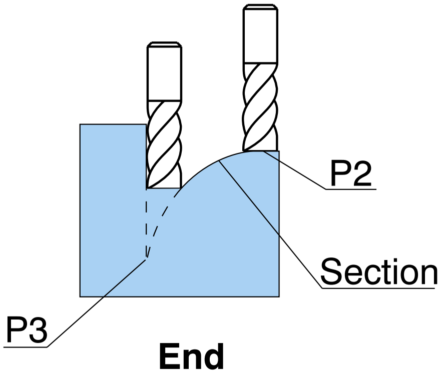
The **Constraint** option enables you to control the tool movement at the start and end of the profile/section. Leave this option unchecked to use no constraints at all.


Both
P2
No
End
Profile
P2
P3
P3
Section


Section
P2
P3
Profile

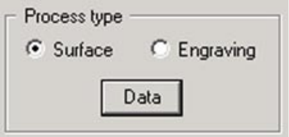

Section
P2
P3
No


P2
P3
Section
Both


P2
Section
P3
End

6.5.7 Process type

The translated surface job can be used to mill surfaces and also to engrave profiles, such as text, on a translated surface.

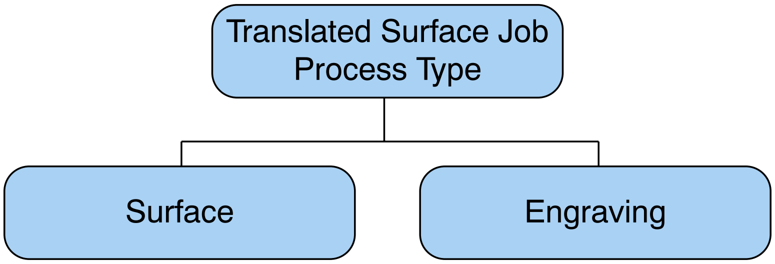


Process type

● Surface ○ Engraving

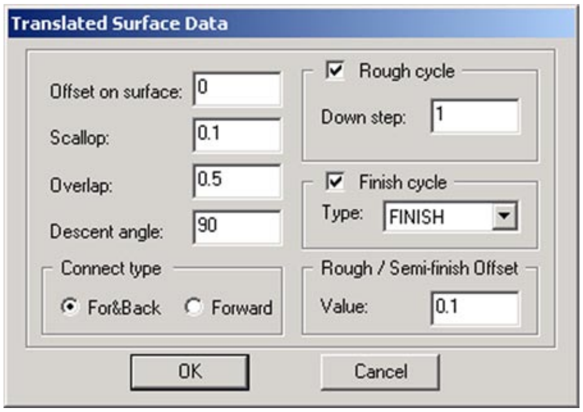
To engrave a profile on a translated surface, you can use the translated surface job that you used to mill the surface as a **Template**. Simply choose **Add Job>Translated Surface** and select the original translated surface job from the **Job Name** list in the Job screen. All the geometries and parameters of the original jobs will be copied.

Enter the **Process type** and click on the **Data** button to define additional parameters.



6.5.8 Surface Data for Translated Surfaces

When you click on the Data button, the **Surface Data dialog** will be displayed.



Translated Surface Data

Offset on surface: 0

Scallop: 0.1

Overlap: 0.5

Descent angle: 90

Connect type

● For&Back ○ Forward

☑ Rough cycle

Down step: 1

☑ Finish cycle

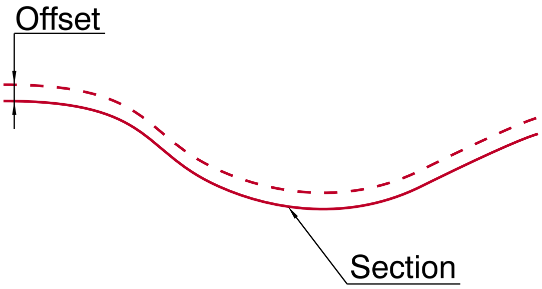
Type: FINISH

Rough / Semi-finish Offset

Value: 0.1

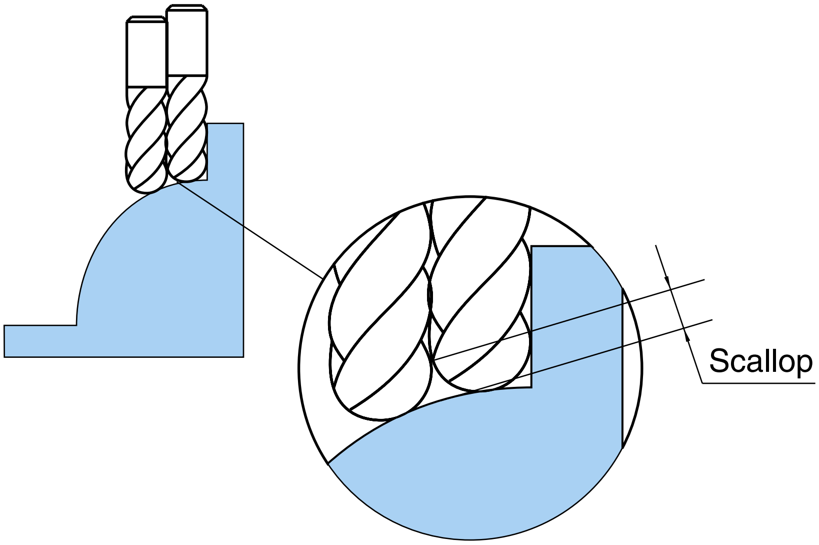
Offset on surface

This value causes the final milled surface to be at an **Offset** from the defined surface.



Scallop

The **Scallop** value determines how smooth the resulting surface finish will be. The smaller the value, the smoother the surface finish will be. The value is used for both the Finish and the Semi-finish operation.



Overlap

This value defines the tool overlap for adjacent tool paths during roughing operations. A value of 0.6 means that the tool will overlap 60% to the next tool path. For a tool diameter of 10 this would result in a side step of 4. Additionally this value defines the maximum side step used in Semi-finish and Finish operations if the calculated scallop would allow a wider side step.

Descent Angle

The tool will plunge into the material with the given ramp angle. The default value of 90 degrees causes a normal approach into the material. The tool enters the material at an angle to the first line or arc of the tool path. If the length required to perform the ramp angle is longer than the first tool path movement, SolidCAM will generate a zig-zag movement to plunge into the material.

Connect type

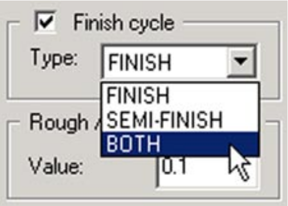
- **For & Back**. The surface is milled with both conventional and climb tool movements.
- **Forward**. The surface is milled only in one direction. This option cannot be used if you use Limits for the translated surface.

Rough cycle

If you check this option, a rough operation will clear the surface area before the semi-finish operation will be performed. Enter the **Down step** between two successive cuts during roughing. Define the remaining offset that will remain on the surface walls and floor after the roughing cut in the field **Rough/Semi-finish offset**.

Finish cycle

Check this option if you intend to semi-finish and/or finish the translated surface. Use the list to select the cycles you want to perform.



☑ Finish cycle

Type: FINISH

Value: 0.1

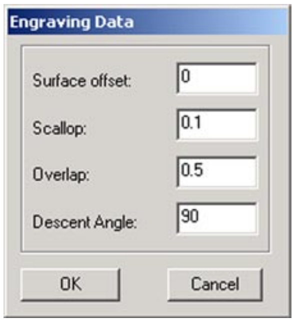
- **Semi-finish** will prepare the slot at an offset specified in the field **Rough/Semi-finish** offset.
- **Finish** will machine the slot to its final dimensions and specified surface scallop.
- **Both**. Use this option to machine the slot with a semi-finish cut first and then a finishing cut.

Rough/Semi-finish offset.

This value specifies the offset that remains on the surface after rough and semi-finish operations. This offset will be removed in the last finish cut.

6.5.9 Engraving Data for Translated Surface Job

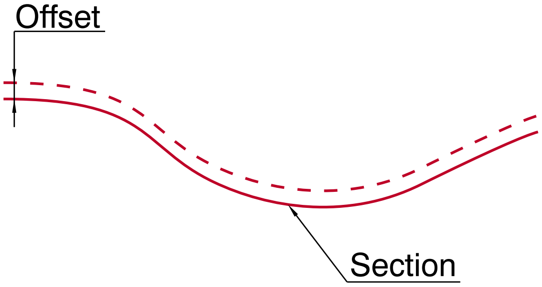
When you click on the **Data** button, the **Engraving Data dialog** will be displayed.



Surface offset: 0
Scallop: 0.1
Overlap: 0.5
Descent Angle: 90

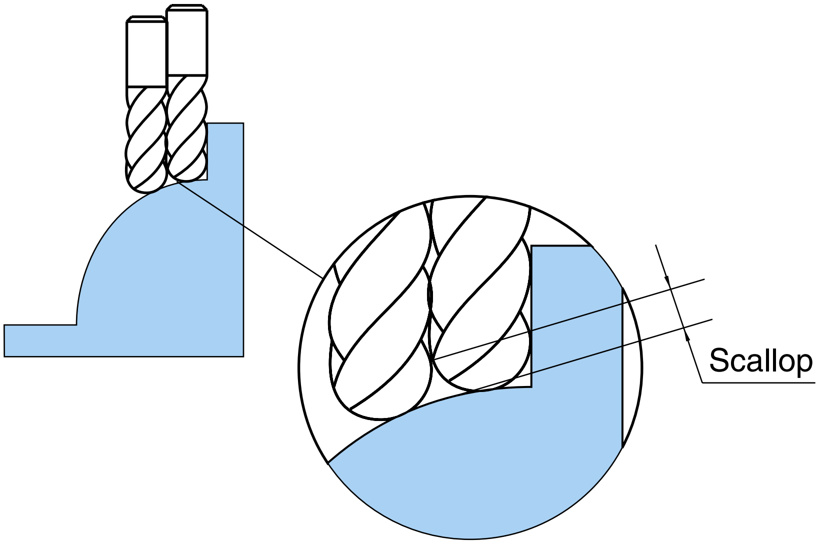
Surface offset

This value causes the final milled surface to be at an **Offset** from the defined surface.



Scallop

The **Scallop** value determines how smooth the resulting surface finish will be. Use smaller values for a smoother surface finish. The value is used for both the Finish and the Semi-finish operations.



Overlap

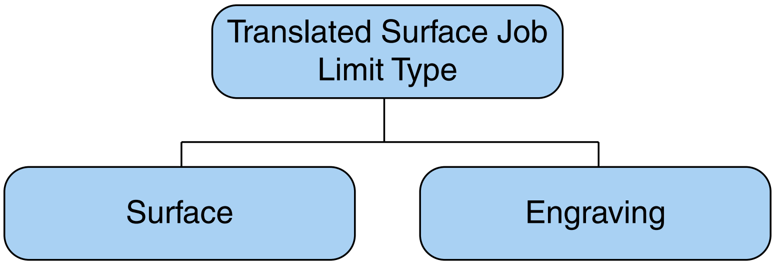
This value defines the tool overlap for adjacent tool paths during roughing operations. A value of 0.6 means that the tool will overlap 60% to the next tool path. For a tool diameter of 10, this would result in a side step of 4. Additionally this value defines the maximum side step used in Semi-finish and Finish operations if the calculated scallop would allow a wider side step.

Descent Angle

The tool will plunge into the material with the given ramp angle. The default value of 90 degrees causes a normal approach into the material. The tool enters the material at an angle to the first line or arc of the tool path. If the length required to perform the ramp angle is longer than the first tool path movement, SolidCAM will generate a zig-zag movement to plunge into the material.

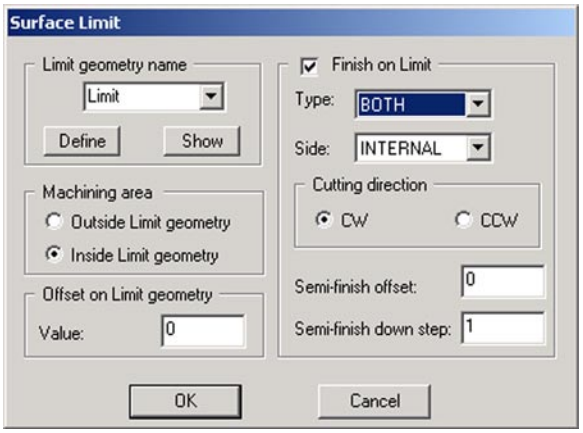
6.5.10 Limits

A limit is a profile geometry that can be used to define additional 2D profiles on the surface and to exclude areas on the surface from machining. If you would like to use a limit on your translated surface, check the **Limits** option and click on the **Data** button. To use the **Engraving process type**, you must specify the **Engraving geometry** and additional parameters in the **Engraving Limit dialog**.



6.5.11 Limit Data for Translated Surface

When you click on the **Data** button, the **Limit Data dialog** will be displayed.



Surface Limit

Limit geometry name: Limit

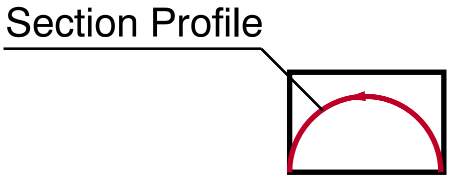
Machining area:
○ Outside Limit geometry
● Inside Limit geometry

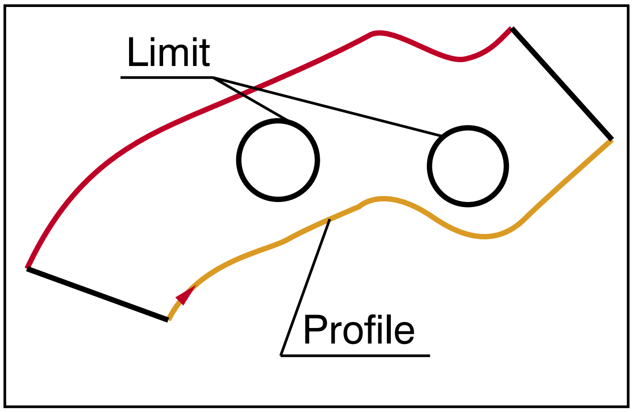
Offset on Limit geometry
Value: 0

☑ Finish on Limit
Type: BOTH
Side: INTERNAL

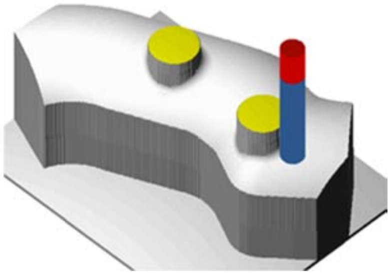
Cutting direction: ● CW ○ CCW

Semi-finish offset: 0
Semi-finish down step: 1





The geometries in the above example will result in this **Translated surface with Limits**.



Geometry name

In this section, specify which contour, i.e. profile geometry you want to use as the Limit. Note that the Limit geometry must be closed.

Machining area

- **Outside Limit geometry.**

  The area of the translated surface outside the limit will be machined.

- **Inside Limit geometry.**

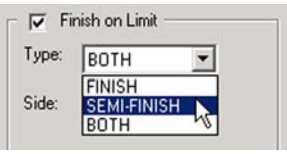
  Only the area inside of the limit geometries will be machined.

Offset on Limit geometry

After machining, the CAM-Part will have remaining material on the limit geometry equal to this value.

Finish on Limit

Check this option if you intend to semi-finish and/or finish the limits on the translated surface. Use the list to select the cycles you want to perform.



☑ Finish on Limit

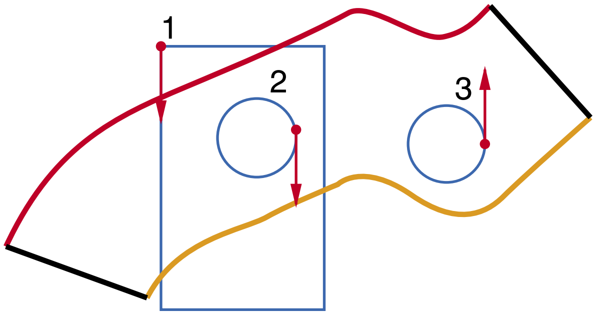
Type: BOTH

- **Finish** will machine the limit to its final dimensions in one down step.
- **Semi-finish** will prepare the limits to an offset specified in the **Offset** field with a cutting depth given in the **Down step** field.
- **Both** - Use this option to machine the limit with a semi-finish cut first and then a finishing cut.

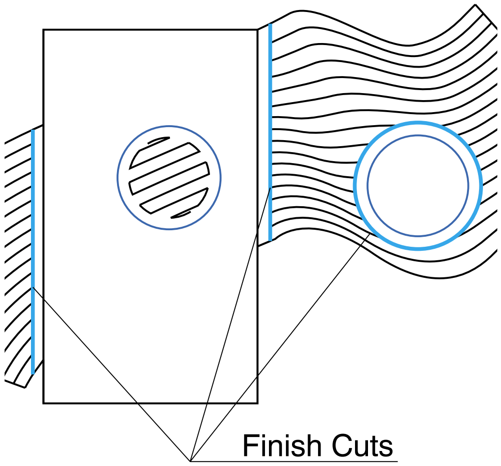
Side

If the Limit geometry is composed of one or more chains, but none of the chains are placed inside another chain, the default option **Both** should be used. SolidCAM will then use the relevant finish side depending on whether the Limit type is Outside or Inside.

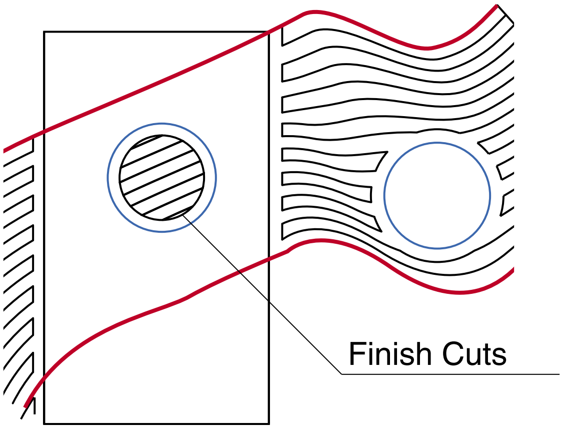
In this Example, a Limit geometry contains a chain inside another chain (Chain #2 inside Chain #1) and a separate Limit chain #3.



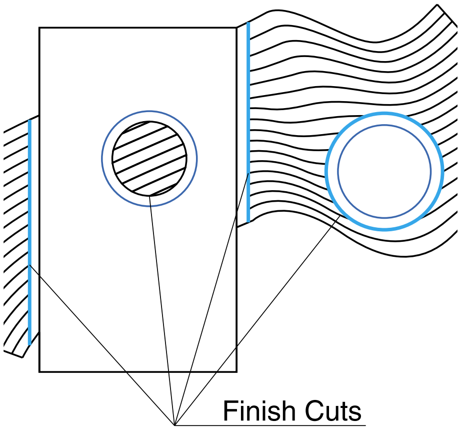
- The **External** option will finish all outside contours of the Limit chains, leaving the inside hole unfinished.



- The **Internal** option will finish only the inside contour of the Limit chain, leaving the outside Limit contours unfinished.



- The **Both** option will finish all inside and outside contours of the Limit chains.



Cutting direction

This field defines the cutting direction on the last finish cut. It can be either clockwise (CW) or counterclockwise (CCW).

Semi-finish offset

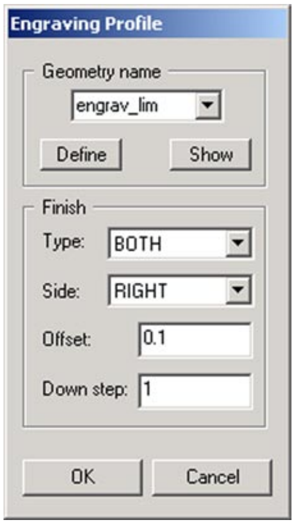
Enter the offset that will remain on the surface after the semi-finish operation. This offset will be removed in a last cut, if a finish cut has been specified.

Semi-finish down step

The down step defines the cutting depth of the tool on the Limit geometries used for the semi-finish operation.

6.5.12 Engraving Limit for Translated Surface Job

When you click on the **Data** button, the **Engraving Limit dialog** will be displayed.



Geometry name: engrav_lim

Finish

Type: BOTH

Side: RIGHT

Offset: 0.1

Down step: 1

Engraving depth

The **depth of the engraved profile** on the translated surface must be entered in the field **Job Upper plane** of the main job screen. If you want to penetrate the translated surface by 1.0, you have to enter a value of -1.0 into the Job upper plane field.

In this **Example**, a text was defined as the Engraving Limit geometry and was engraved on a translated surface.

CADTECH GMBH

Engraving geometry

In this section you have to specify the contour, i.e. profile geometry, which you want to engrave on the translated surface.

Finish options

Type

This option enables you to semi-finish and/or finish the engraved profile on the translated surface.

- **Semi-finish** will prepare the engraving profile to an offset specified in the Offset field with cutting depths given in the Down step field.
- **Finish** will machine the engraving profile to its final dimensions in one down step.
- **Both**. Use this option to machine the limit with a semi-finish cut first and then a finishing cut.

Side

The tool can cut on the left/right to the profile or on the engraving geometry.

- **Right**. The tool will cut on the right side of the profile geometry up to a distance defined by the Offset value.
- **Left**. The tool will cut on the left side of the profile geometry at a distance defined by the Offset value.
- **Middle**. The center of the tool will move on the profile geometry.

Offset

You can specify the offset from the engraving geometry for a Semi finish operation on the left or right side of the geometry. This offset will be removed by the Finish cut, if you had defined one.

Down step

This value defines the successive Z-cutting depth during the Semi-finish operation of the engraving profile.

6.5.13 Approach/Retreat

You can control the way the tool approaches and retreats from the geometry. Select an option from the list field and enter the **Distance value** for this option.

SolidCAM enables you to use the following **Approach/Retreat** options

- **None**
- **Normal**
- **Arc**
- **Tangent**

For more information on this subject, please refer to topic **6.1.12**.

6.5.14 Show surface

You can display the geometries and the tool to check the current settings for the profile direction, tool side etc...

3D Milling

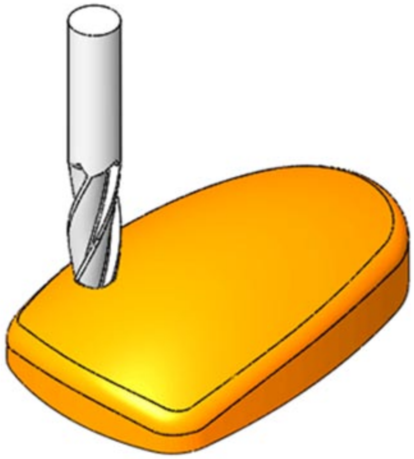
7

SolidCAM's 3D Milling can be used both for prismatic parts and for complex 3D models. For prismatic parts SolidCAM analyzes the model and automatically recognizes pockets and profiles to be machined using Zconstant machining strategies.

For molds, electrodes and prototypes, SolidCAM offers powerful 3D machining, including advanced High Speed Machining strategies and integrated rest material options. No matter how complex the model, SolidCAM provides the optimal approach and roughing strategy with superior 3D finish machining for mold and die applications.

❒ 3D Model Job
❒ Rough Strategies
❒ Semi-Finish & Finish Strategies
❒ 3D Engraving Job
❒ 3D Drill Job

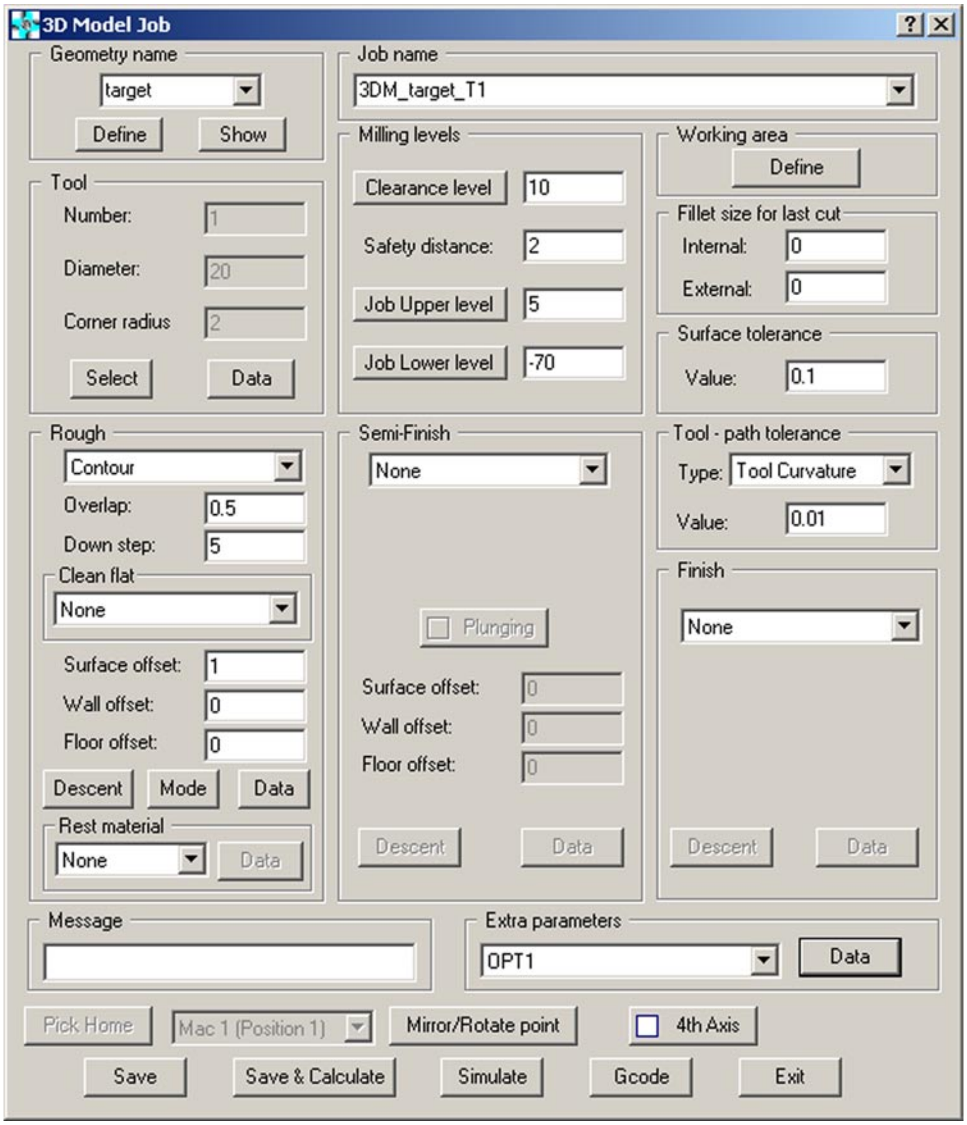
7.1 3D Model Job Overview



You can perform 3-axis gouge-free machining on solid and surface models. This Job offers a wide range of roughing, semi-finishing and finishing strategies for free-form models. It can be used to manufacture molds, dies, electrodes, prototypes and other **3D Models**.

When you select the commands **Add>3D Model**, the **3D Model Job** screen will be displayed.

Select the 3D model geometry for the Job from the list.



3D Model Job
Define
Show
Select
Data
Clearance level
Job Upper level
Job Lower level
Define
Descent
Mode
Data
Data
Descent
Data
Descent
Data
Data
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

Geometry name: target

Job name: 3DM_target_T1

Tool

Number: 1

Diameter: 20

Corner radius: 2

Milling levels

Clearance level: 10

Safety distance: 2

Job Upper level: 5

Job Lower level: -70

Working area

Fillet size for last cut

Internal: 0

External: 0

Surface tolerance

Value: 0.1

Rough: Contour

Overlap: 0.5

Down step: 5

Clean flat: None

Surface offset: 1

Wall offset: 0

Floor offset: 0

Rest material: None

Semi-Finish: None

☐ Plunging

Surface offset: 0

Wall offset: 0

Floor offset: 0

Tool - path tolerance

Type: Tool Curvature

Value: 0.01

Finish: None

Message:

Extra parameters: OPT1

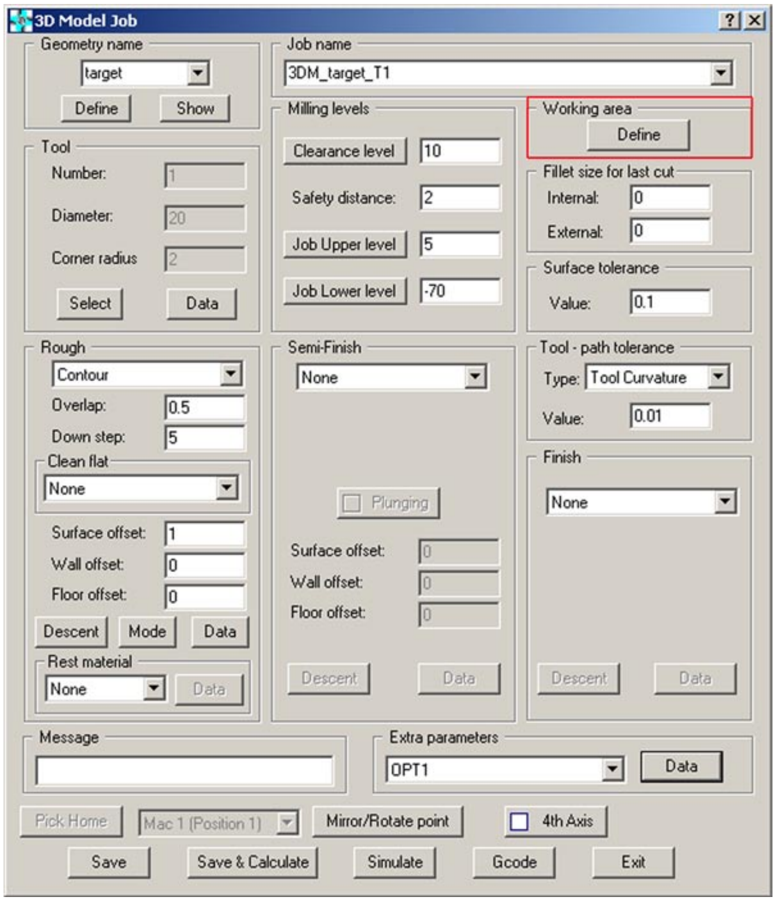
Mac 1 (Position 1)

☐ 4th Axis

7.2 Working Area

SolidCAM enables you to limit the tool path with a specified working area. In case a Working area is used, SolidCAM calculates the whole tool path and then removes the all tool path lines that are not inside the Working area.

In the **Working area** field, click on the **Define button**; the **Working area dialog** will be displayed.


3D Model Job

Geometry name: target

Job name: 3DM_target_T1

Tool

Number: 1

Diameter: 20

Corner radius: 2

Milling levels

Clearance level: 10

Safety distance: 2

Job Upper level: 5

Job Lower level: -70

Working area

Fillet size for last cut

Internal: 0

External: 0

Surface tolerance

Value: 0.1

Rough: Contour

Overlap: 0.5

Down step: 5

Clean flat: None

Surface offset: 1

Wall offset: 0

Floor offset: 0

Rest material: None

Semi-Finish: None

☐ Plunging

Surface offset: 0

Wall offset: 0

Floor offset: 0

Tool - path tolerance

Type: Tool Curvature

Value: 0.01

Finish: None

Message

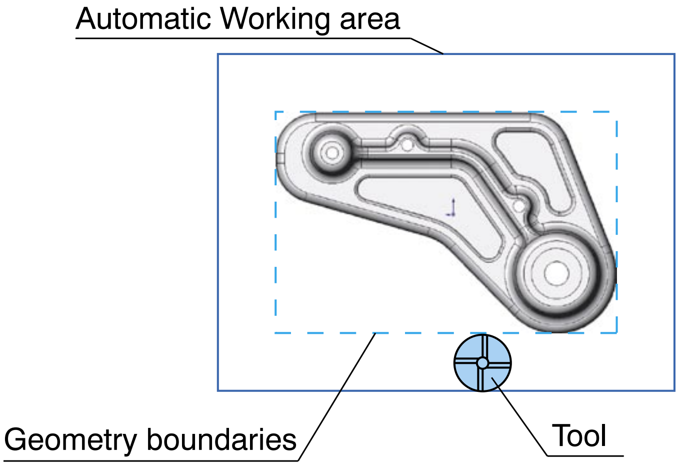
Extra parameters: OPT1

Mac 1 (Position 1)

☐ 4th Axis

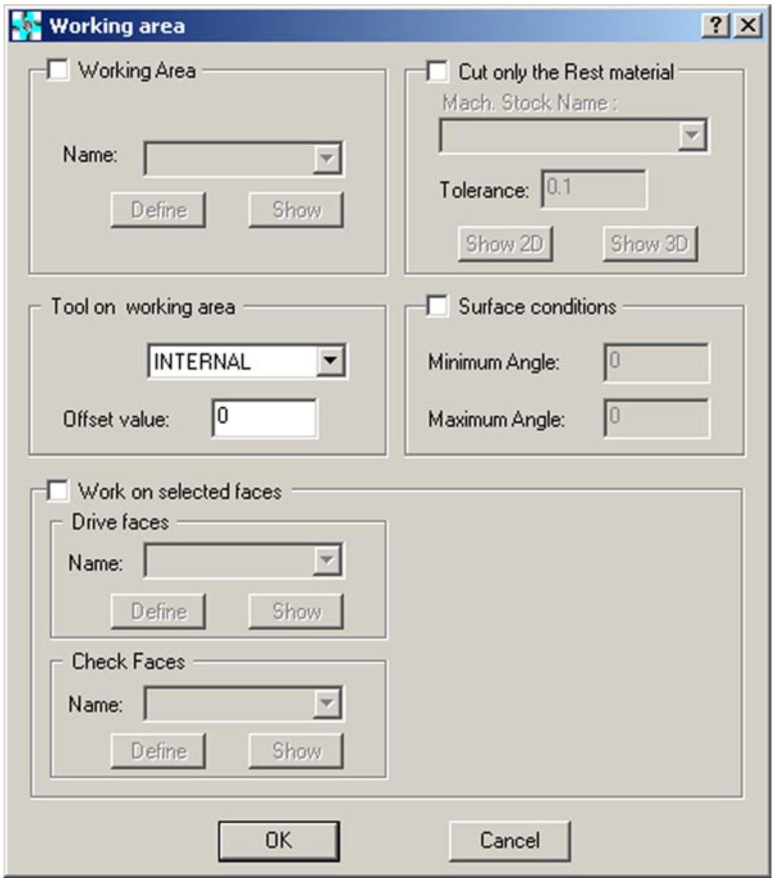
?
Define
Show
Select
Data
Define
Descent
Mode
Data
Data
Descent
Data
Descent
Data
Data
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

If you do **NOT** define a working area, a box at an offset with the width of the tool diameter will be drawn around the current 3D model. SolidCAM will apply the selected milling strategy to the complete Automatic Working Area.


Automatic Working area
Geometry boundaries
Tool

7.2.1 Working area dialog

The **Working area dialog** enables you to choose the type of working area and set the Working area parameters.


Working area
? ✕
☐ Working Area
Name:
Define
Show
☐ Cut only the Rest material
Mach. Stock Name :
Show 2D
Show 3D
☐ Surface conditions
☐ Work on selected faces
Drive faces
Name:
Define
Show
Check Faces
Name:
Define
Show
OK
Cancel

Tolerance: 0.1

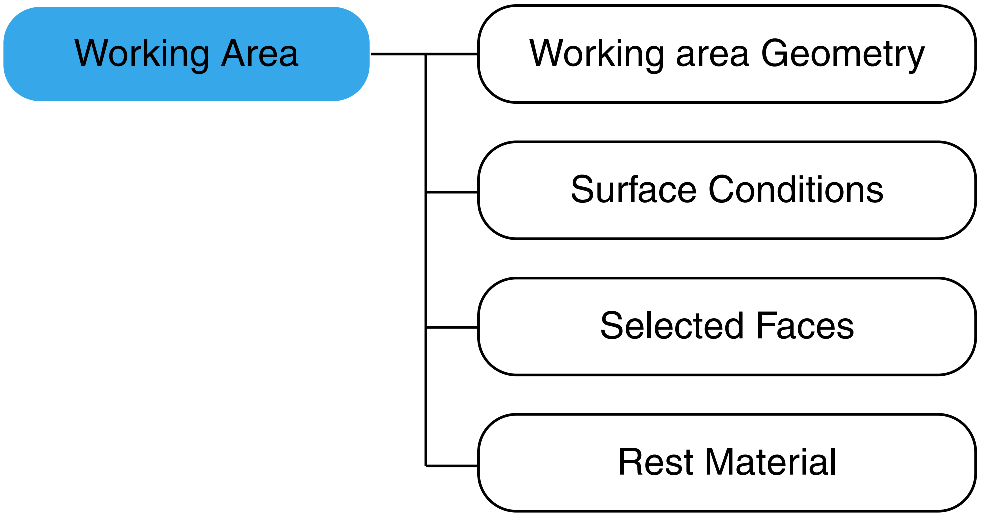
Tool on working area: INTERNAL

Offset value: 0

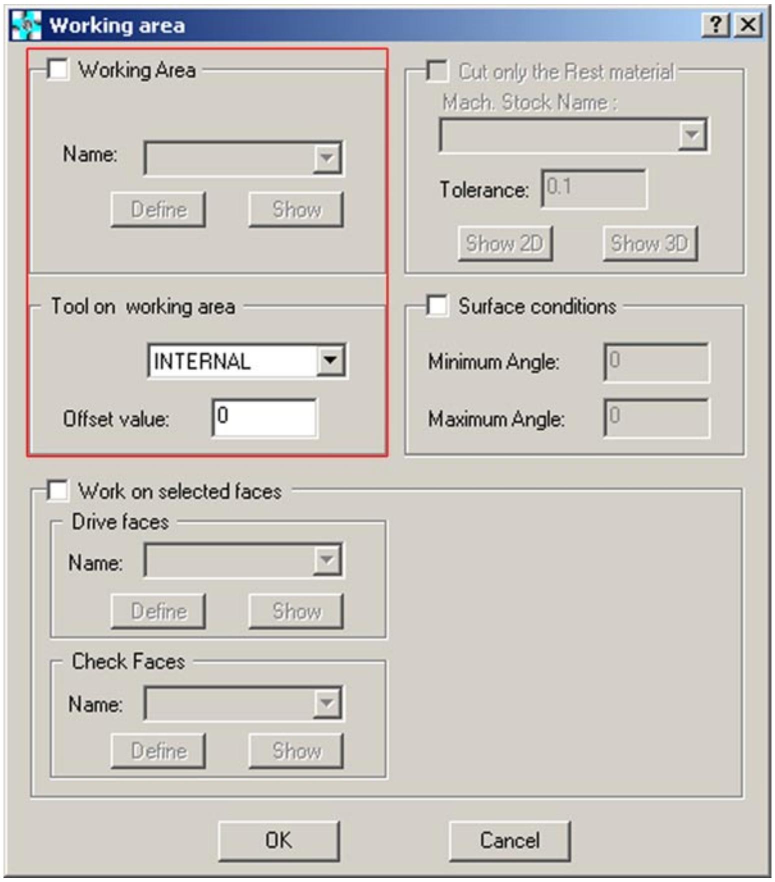
Minimum Angle: 0

Maximum Angle: 0

SolidCAM enables you to use either one or a combination of the following three methods:


Working Area
Working area Geometry
Surface Conditions
Selected Faces
Rest Material

7.2.2 Working area definition via Geometry


Working area
☐ Working Area
Name:
Define
Show
Tool on working area
Offset value:
☐ Cut only the Rest material
Mach. Stock Name :
Tolerance:
Show 2D
Show 3D
☐ Surface conditions
Minimum Angle:
Maximum Angle:
☐ Work on selected faces
Drive faces
Name:
Define
Show
Check Faces
Name:
Define
Show
OK
Cancel

Tool on working area: INTERNAL

Offset value: 0

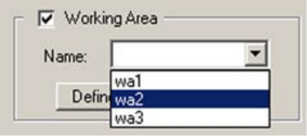
Tolerance: 0.1

Minimum Angle: 0

Maximum Angle: 0

In this section you have to specify which working area geometry defines the machining boundaries for this job.

If you have already defined working area geometries for this CAM-Part, you can select a geometry from the **Geometry name** list field.

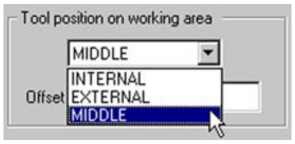

☑ Working Area
Name:
Defin
wa1
wa2
wa3

- **Show** will display the selected geometry in the Host CAD window.
- **Define** enables you to select a new working area geometry. (For more information on the Geometry selection, please refer to section ...)

Tool on working area

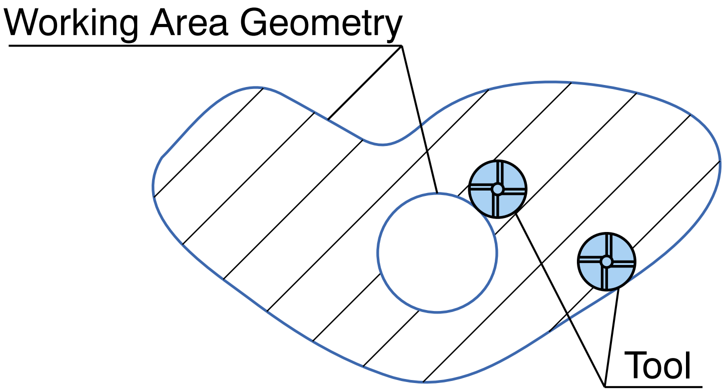
This option controls how the tool is positioned relative to the boundaries of the working area. Since the working area geometry is always projected on a 2D plane, viewing the model and working area in the top view shows exactly till where the tool will machine the model.

Select one of the options from the **Tool on working area** list field.


Tool position on working area
MIDDLE
INTERNAL
EXTERNAL
MIDDLE
Offset

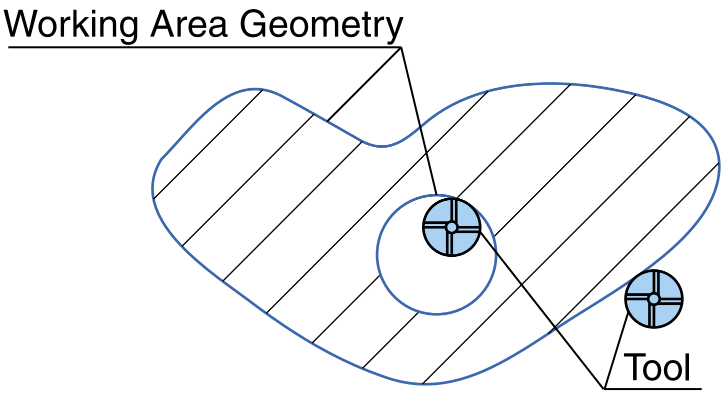
- **Internal**

  The tool machines up to the boundary of the working area. The center point moves at an offset equal to the tool radius.


Working Area Geometry
Tool

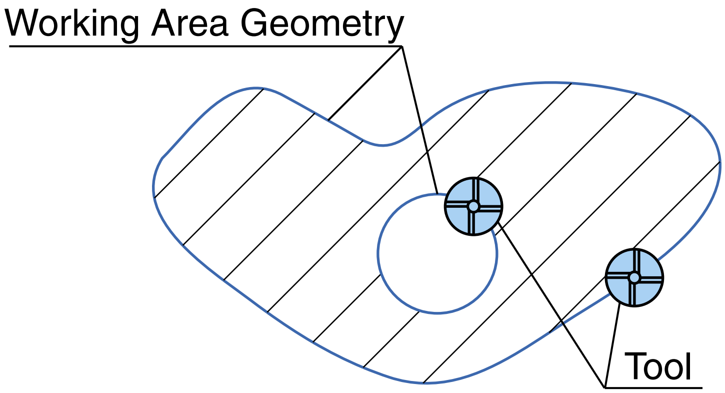
- **External**

  The tool machines over the boundary of the working area by a distance equal to the tool diameter.


Working Area Geometry
Tool

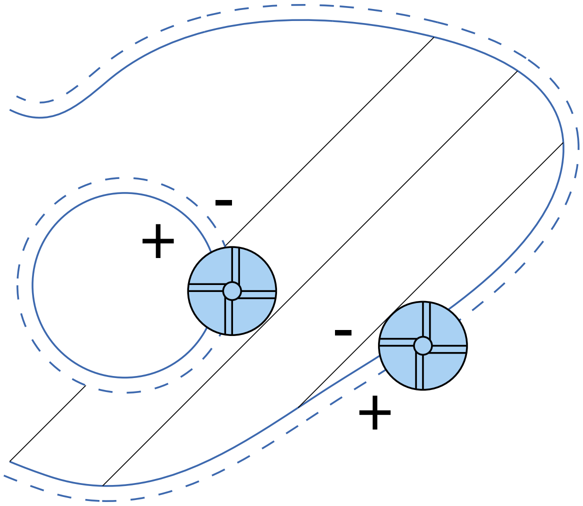
- **Middle**

The tool machines the working area until the tool center is positioned on the working area.



- **Offset value (+/-)**

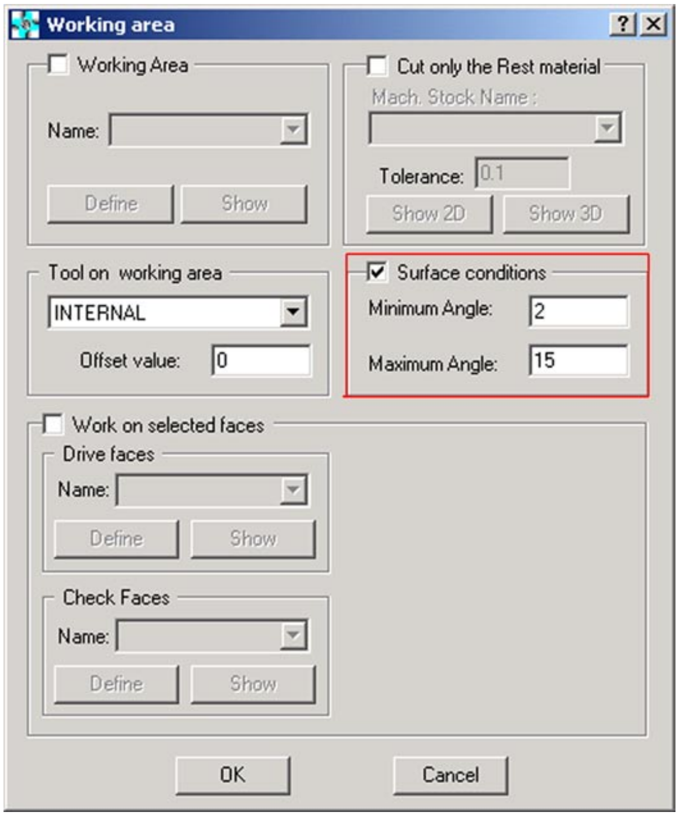
This value enables you to specify the offset of the tool center.



A positive offset value will enlarge the working area; a negative value will reduce the working area to be machined.

- To stay away from a pocket's wall, for example, you can use the internal option with a small negative offset value.
- To avoid a tool plunging down a straight wall of a block, you can use the **Middle** option with offset less than tool Radius. Also, with the same result, you can use the External option with a small negative offset value.

7.2.3 Working area definition via Surface condition


Working area
Define
Show
Show 2D
Show 3D
Drive faces
Check Faces
Define
Show
OK
Cancel

☐ Working Area
Name:

☐ Cut only the Rest material
Mach. Stock Name:
Tolerance: 0.1

Tool on working area: INTERNAL
Offset value: 0

☑ Surface conditions
Minimum Angle: 2
Maximum Angle: 15

☐ Work on selected faces
Drive faces – Name:
Check Faces – Name:

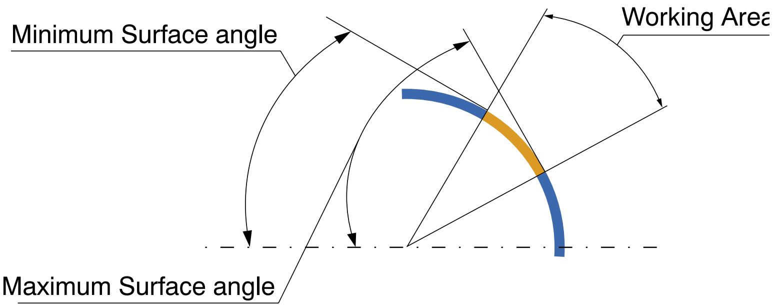
Choose this option if you want to apply milling strategies to areas inside a specific angle range. You could, for example, machine all areas inside a working area on the model, that have an angle from 0 to 30 degrees with a Linear finish strategy. All areas from 30 to 90 degrees could later be machined in a Constant-Z job.

- **Minimum surface angle**

  This value defines the lower limit of the angle range. All areas that are more flat than this angle will not be included in the working area.

- **Maximum surface angle**

  All areas on the model or inside the user-defined working area that are steeper than this angle will be excluded from machining.


Minimum Surface angle
Working Area
Maximum Surface angle

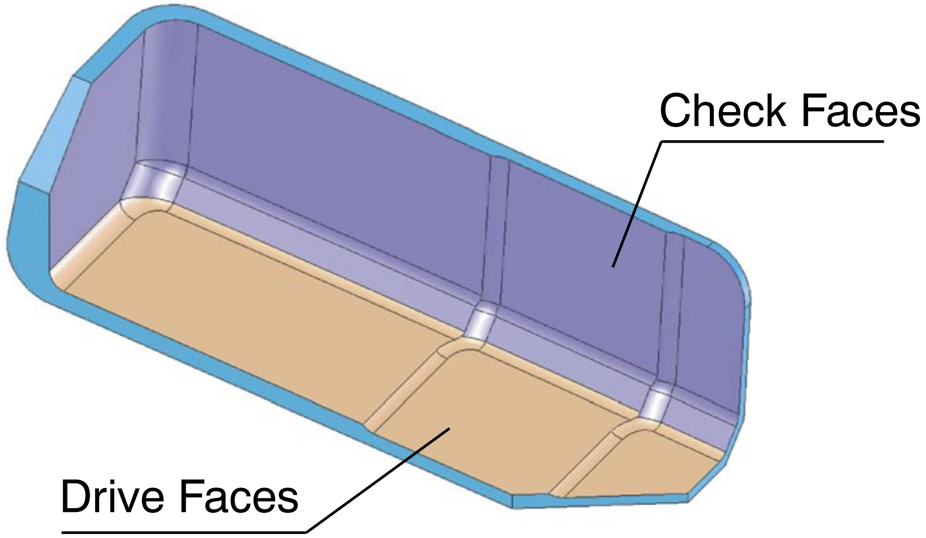
7.2.4 Working area definition via Face selection

SolidCAM enables you to define the area of the 3D model machining with two sets of model faces:

- **Drive faces** – the set of faces to be milled. The tool path will be generated only for machining of these faces.
- **Check faces** – the set of faces to be avoided during the generation of the tool path.

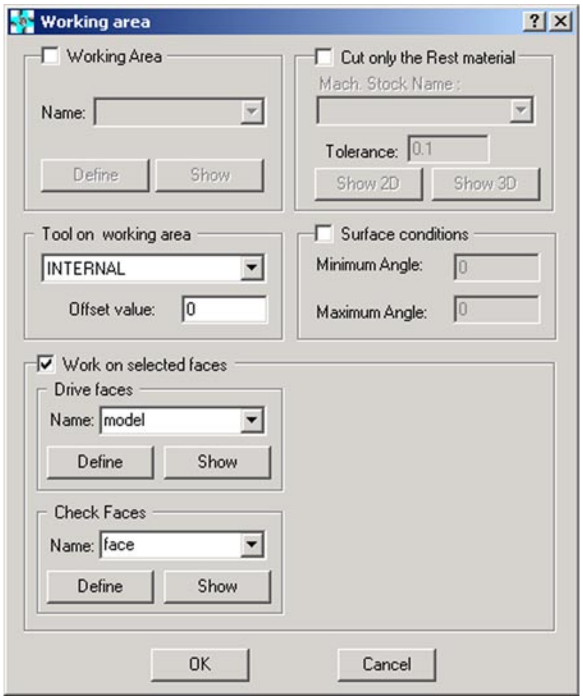
Other faces of the model will not be considered during the tool path calculation.

For example:



Within the model shown above, SolidCAM generates the tool path only for the pocket area (green). During the calculation, SolidCAM avoids the gouging of the check faces (orange). The remaining faces will not participate in the tool path calculation.

The above capability is available through the **Work on selected faces** area in the **Working area dialog**.



☐ Working Area
Name:

☐ Cut only the Rest material
Mach. Stock Name:
Tolerance: 0.1

Tool on working area: INTERNAL
Offset value: 0

☐ Surface conditions
Minimum Angle: 0
Maximum Angle: 0

☑ Work on selected faces
Drive faces
Name: model
Check Faces
Name: face

Drive faces

Specify the **Face Geometry** of **Drive faces**. If you have already defined such geometries for this CAM-Part, you can select the geometry from the **Name** list field.

- The **Define** button enables you to define a new **Face Geometry** with the **Choose Faces dialog**.
- The **Show** button will display the **Face Geometry** on the SolidWorks model.

Check faces

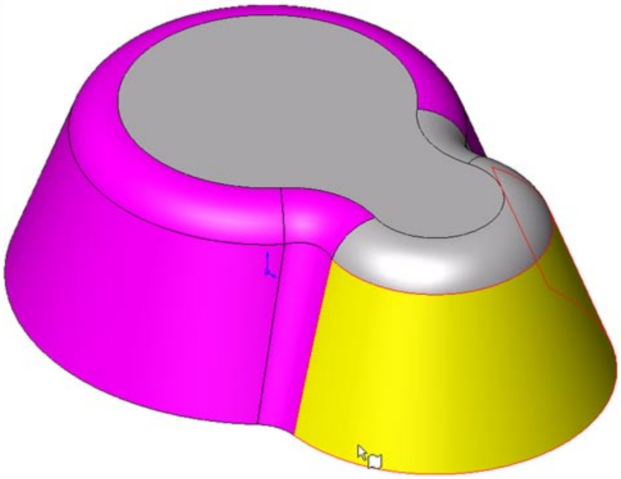
Specify the **Face Geometry** of Check faces. If you have already defined such geometries for this CAM-Part, you can select the geometry from the **Name** list field.

- The **Define** button enables you to define a new **Face Geometry** with the **Choose Faces dialog**.
- The **Show** button will display the **Face Geometry** on the SolidWorks model.

Choose Faces dialog

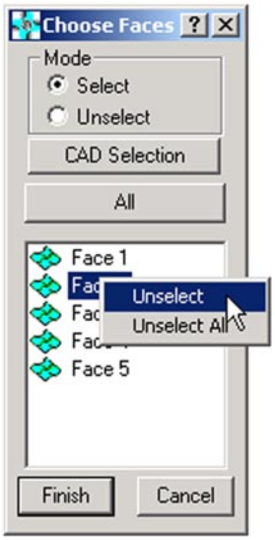
This dialog enables you to select one or several faces of the SolidWorks model. The selected Face tags will be displayed in the dialog.

Choose Faces
Mode
Select
Unselect
CAD Selection
All
Face 1
Face 2
Face 3
Face 4
Face 5
Finish
Cancel

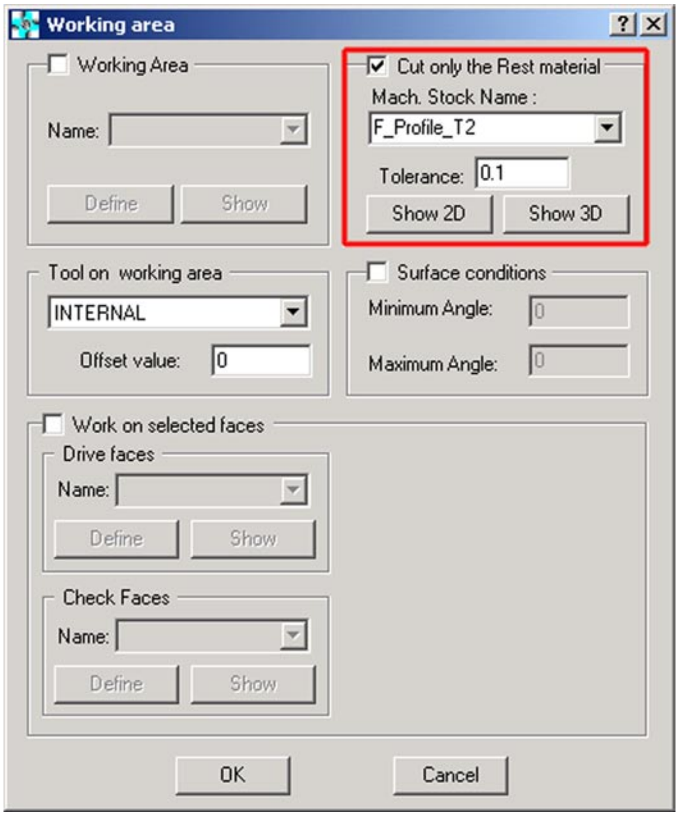


If you chose wrong entities, use the **Unselect** option to undo your selection. You can also right click on the entity name (the object will be highlighted) and choose the **Unselect** option from the menu.

The **CAD Selection** option enables you to select faces with the host CAD tools. The **All** option enables you to select all the model faces.


Choose Faces
Mode
Select
Unselect
CAD Selection
All
Face 1
Fac
Unselect
Unselect All
Fac
Fac
Face 5
Finish
Cancel

7.2.5 Working area definition via Rest material



☐ Working Area

Name:

☑ Cut only the Rest material

Mach. Stock Name : F_Profile_T2

Tolerance: 0.1

Tool on working area: INTERNAL

Offset value: 0

☐ Surface conditions

Minimum Angle: 0

Maximum Angle: 0

☐ Work on selected faces

Drive faces

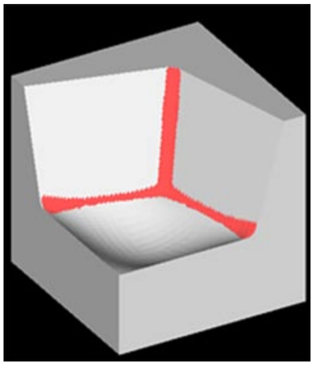
Name:

Check Faces

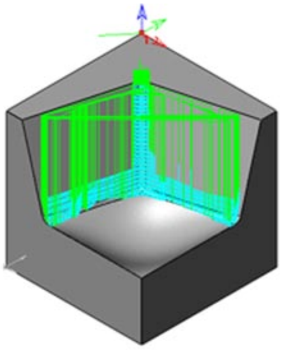
Name:

The Rest material is what has to be machined after the previous operations in order to reach the Target model. SolidCAM automatically compares the updated material after each job stock model and the target model and generates the rest material areas.

With the **Cut only in rest material** option you can limit the machining to the rest material.



The rest material left after semi-finishing operations with the big tool.



Working only in the rest material area.

This option will work only if the following conditions are defined in the **CAM-Part data dialog**.

- The **target model** is defined
- The **Update Stock model** check box is marked.

Mach. Stock Name

This option enables you to use the previously generated Stock model saved in the STL format for the rest material calculation.

This field is active only when the **Solid Rest material calculation** method and **Manual** method of the **Rest material update** are chosen. (For more information on this subject, please refer to the topic **2.4**.)

Show2D

This will show you a 2D boundary around the area where there is material left over.

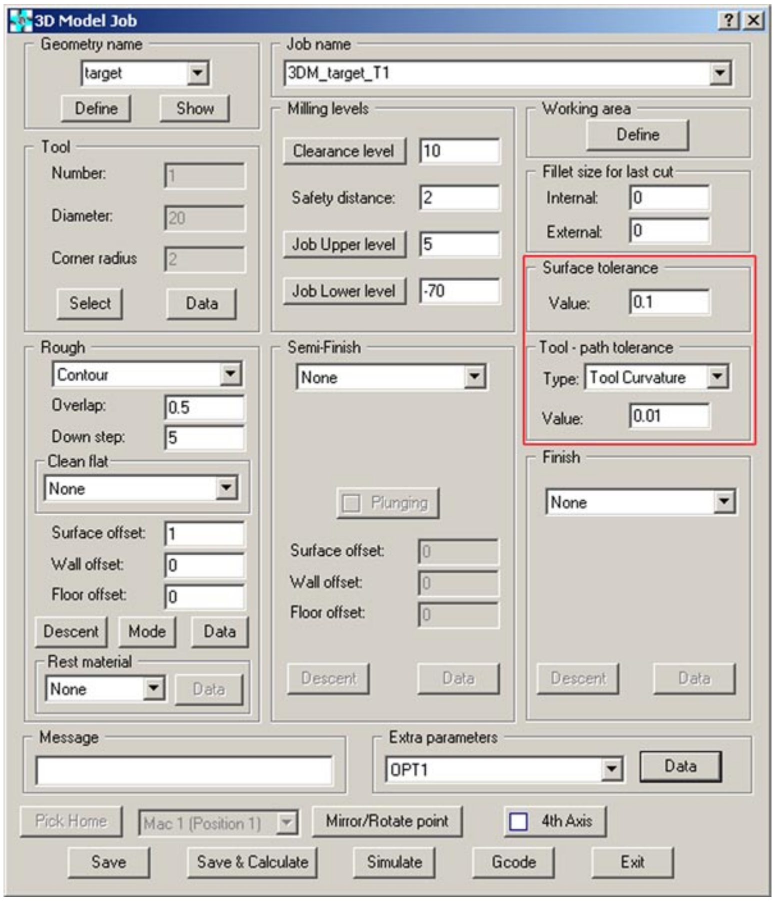
Show3D

This option will show the actual material that is left on the 3D target.

Tolerance

The tolerance of the working area (via rest material) is a visual tool that has no effect on the actual machining.

7.3 Tolerance


3D Model Job
Define
Show
Select
Data
Clearance level
Job Upper level
Job Lower level
Working area
Define
Plunging
Descent
Mode
Data
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

Geometry name: target

Job name: 3DM_target_T1

Tool

Number: 1

Diameter: 20

Corner radius: 2

Milling levels

Clearance level: 10

Safety distance: 2

Job Upper level: 5

Job Lower level: -70

Fillet size for last cut

Internal: 0

External: 0

Surface tolerance

Value: 0.1

Tool - path tolerance

Type: Tool Curvature

Value: 0.01

Rough: Contour

Overlap: 0.5

Down step: 5

Clean flat: None

Surface offset: 1

Wall offset: 0

Floor offset: 0

Rest material: None

Semi-Finish: None

Surface offset: 0

Wall offset: 0

Floor offset: 0

Finish: None

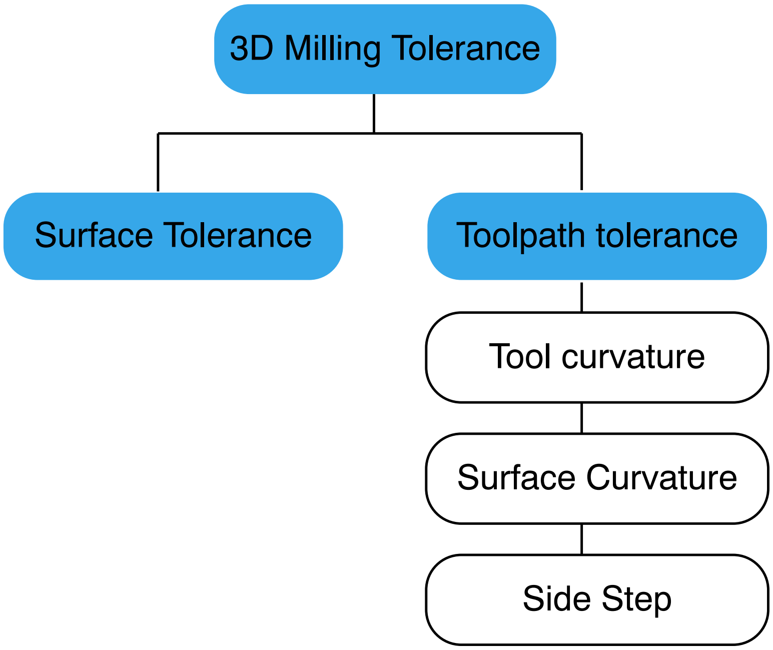
Message:

Extra parameters: OPT1

Mac 1 (Position 1)

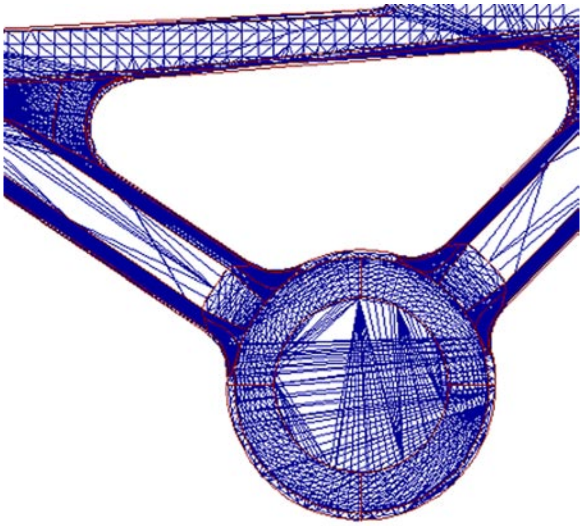
☐ 4th Axis

The precision tolerances defined here will directly affect the surface quality of your model.


3D Milling Tolerance
Surface Tolerance
Toolpath tolerance
Tool curvature
Surface Curvature
Side Step

7.3.1 Surface tolerance

This tolerance controls the maximum deviation of the **mathematical representation** from the original solids and surfaces of your model.



The 3D model geometry will be triangulated and the resulting facets will be saved in the file **modelname.fct** in the CAM-Part‘s folder. The triangulation is performed on the 3D model geometry when you use it for the first time in a 3D model job. If you use the 3D geometry in another job, SolidCAM will check the tolerance of the existing *.fct file of this geometry. It will not perform another triangulation as long as the *.fct file has been created within the same surface tolerance.

7.3.2 Tool path tolerance

SolidCAM generates the tool path in two steps:

1. SolidCAM generates a mathematical representation of the surfaces and solids of the 3D model geometry inside the given **surface tolerance**. The information of this triangulation process will be stored in an internal *.fct file that is similar to an STL file.

2. SolidCAM calculates the tool path using the parameters that have been specified for the tool and the milling strategy to calculate the single steps of the tool on the mathematical model created in the first step. The result of this calculation, the *.pj p-code file, later serves as input to the generation of the actual G-Code file. During this calculation SolidCAM uses the Tool path tolerance that affects on number of single steps in the tool path (accuracy of the tool path).

The tool path tolerance can be defined by the following criteria:

Tool Curvature

With this option the accuracy of the tool path depends on the Tool diameter and the specified tolerance value.

Surface Curvature

With this option the accuracy of the tool path depends on the surface curvature and the specified tolerance value.

Side Step

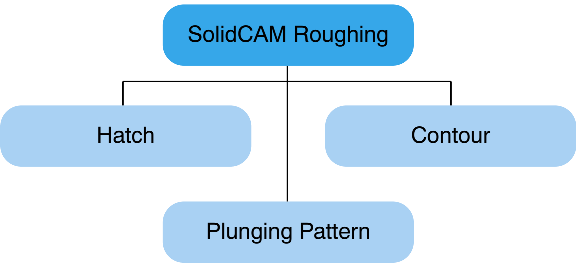
This tolerance enables you to control the size of a single step that will be constant during the whole tool path.

A small tolerance values will result in more G-Code steps in the G-Code file, but the resulting surface will be closer to the mathematical model.

7.4 Roughing

Rough milling removes the bulk of material surrounding the 3D model. The material will be removed on Z-levels controlled by the Job upper plane, surface offset and down step values. To remove large amounts of material, large tools, small feeds and low tolerances are usually used.

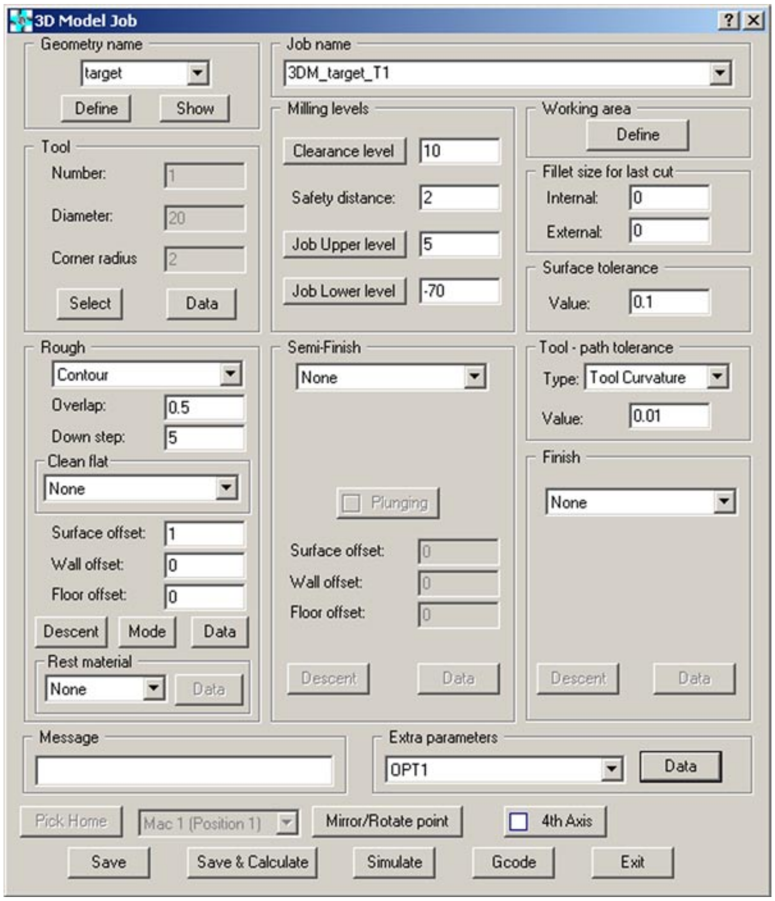
You can use three different strategies to rough machine 3D models.


SolidCAM Roughing
Hatch
Contour
Plunging Pattern

Contour and **Hatch** strategies perform rough cuts on constant Z-levels that are automatically calculated using the specified values for the Job Lower plane, Surface Offset and Down step.

Plunging is a totally different concept of removing material from a given pocket, carried out with a special tool. Instead of milling the material, the tool moves up and down as in a drilling motion, traveling along the specified path type.

The strategy and technological parameters are all defined in the **Rough section** of the 3D model job. Select one of the strategies and specify additional parameters.


3D Model Job
Geometry name
Define
Show
Tool
Select
Data
Milling levels
Clearance level
Job Upper level
Job Lower level
Working area
Define
Fillet size for last cut
Surface tolerance
Rough
Clean flat
Descent
Mode
Data
Rest material
Data
Semi-Finish
Plunging
Descent
Data
Tool - path tolerance
Finish
Descent
Data
Message
Extra parameters
Data
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

Geometry name: target

Job name: 3DM_target_T1

Tool Number: 1
Diameter: 20
Corner radius: 2

Clearance level: 10
Safety distance: 2
Job Upper level: 5
Job Lower level: -70

Fillet size for last cut Internal: 0
External: 0

Surface tolerance Value: 0.1

Rough: Contour
Overlap: 0.5
Down step: 5
Clean flat: None
Surface offset: 1
Wall offset: 0
Floor offset: 0
Rest material: None

Semi-Finish: None
☐ Plunging
Surface offset: 0
Wall offset: 0
Floor offset: 0

Tool - path tolerance Type: Tool Curvature
Value: 0.01

Finish: None

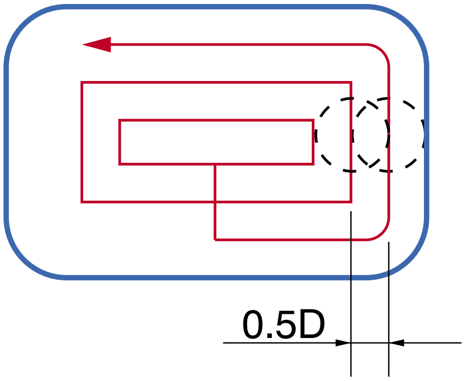
Extra parameters: OPT1

Mac 1 (Position 1)

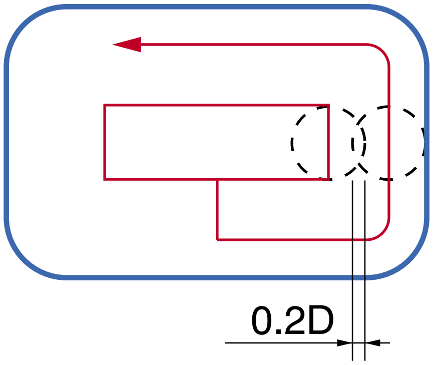
☐ 4th Axis

7.4.1 Overlap

Enter the **Overlap** of adjacent tool paths. The default value is 0.5 (50%). With a tool diameter 10, an overlap of 0.7 would mean a side step of 3.0 for the next tool movement. Values smaller than 0.3 should be avoided.



Overlap 0.5D

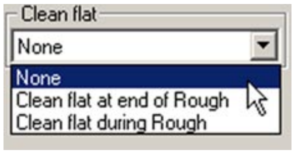


Overlap 0.2D

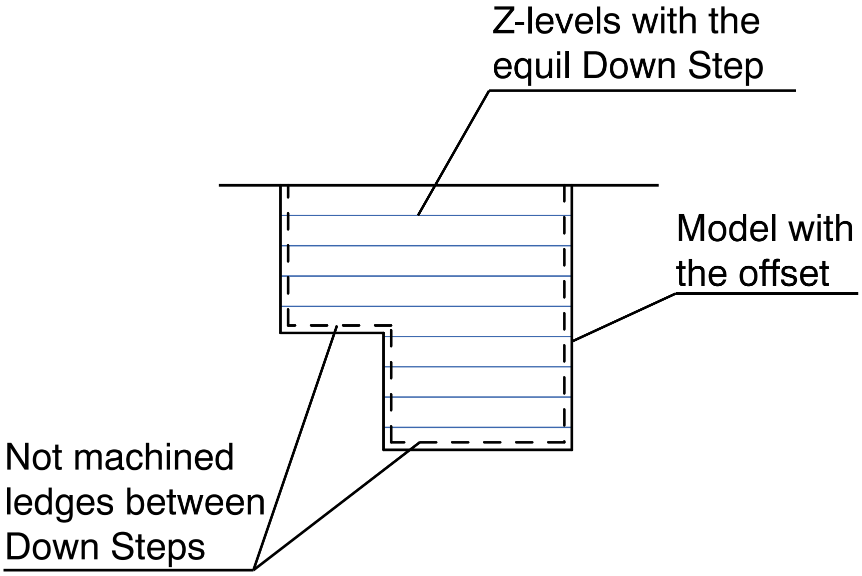
7.4.2 Down step

SolidCAM uses the constant-Z pocketing for the roughing. With the **Down Step** parameter you can define the distance between each two successive Z-levels. Enter the tool's cutting depth.

7.4.3 Clean flat


Clean flat
None
None
Clean flat at end of Rough
Clean flat during Rough

SolidCAM generates a set of Z-levels with equal Down steps. Sometimes the model ledge (with the specified offset) falls between the Down Steps and can't be machined in the actual operation.


Z-levels with the
equil Down Step
Model with
the offset
Not machined
ledges between
Down Steps

- **None**

  If this option is chosen, The tool will machine only at each down step distance. Anything in between will not be machined.

- **Clean flat at end of Rough**

  When this option is chosen, SolidCAM produces complete roughing at each down step level and then returns to the flat areas that are between the down steps.

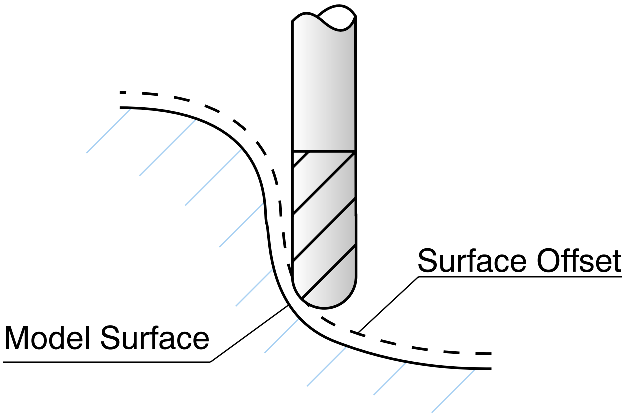
- **Clean flat during roughing**

  When this option is chosen, SolidCAM produces tool paths at the down step level. At this point the tool will machine on the flat area that is located between the down steps. After completing the flat area, it will then go to the next down step.

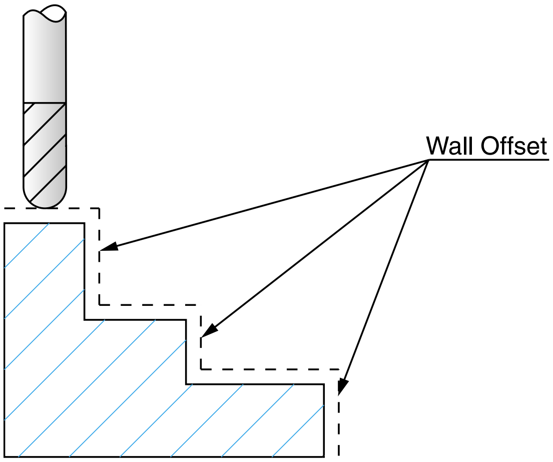
7.4.4 Offset

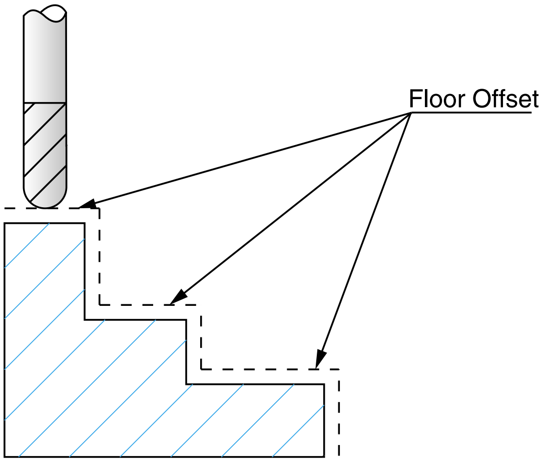
- **Surface offset** - a minimum offset of material in the X, Y and Z direction will remain on the model contour. The value defines the closest distance the tool can get to the model. The rough operation is performed at constant Z-levels.

In addition to the specified **Surface offset**, the roughed model will show stairs of rest material with a height equal to the down step value. To obtain a uniform offset on the model, a semi-finish operation should be executed.



- In 3D milling of prismatic parts we often need to use the different offset values for vertical and horizontal areas. With the **Wall Offset** value you can define the XY offset of any surface that is not absolutely flat. To define the Z offset to the absolute horizontal (flat) surfaces, use the **Floor Offset** parameter.





Use the **Wall** and **Flat offsets** for 3D machining of prismatic parts. In 3D milling of non-prismatic parts, it is best to use surface offset and not floor and wall offsets. Set the **Wall** and **Flat offsets** to **0** and enter the desired value of the **Surface offset**.

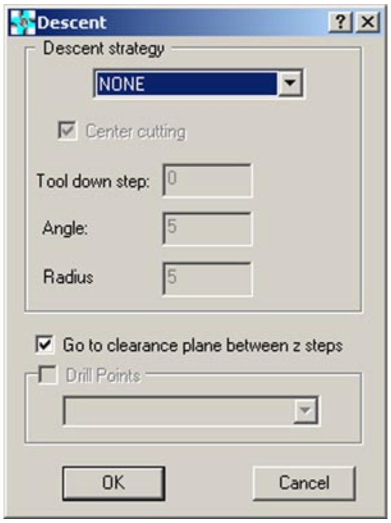
7.4.5 Fillet size for last cut

This option enables you to add a radius to a sharp corners of the tool path without having to change the geometry.

For more information on this subject, please refer to the topic **6.1.4**

7.4.6 Descent

You can control how the tool will plunge into the material. Click on the **Descent** button to display the **Descent dialog**.



Descent strategy: NONE

☑ Center cutting

Tool down step: 0

Angle: 5

Radius: 5

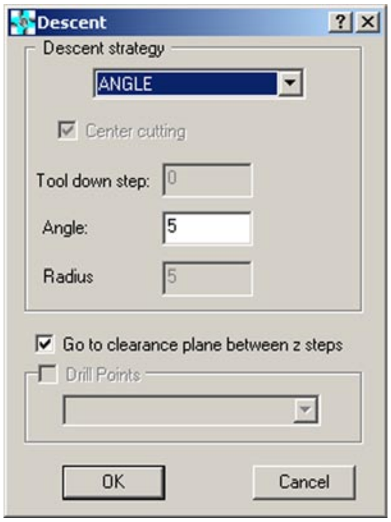
☑ Go to clearance plane between z steps

☐ Drill Points

None

With this type of approach the tool will plunge into the material with a normal downward movement at the specified Z-feed rate.

Angle



Descent strategy: ANGLE

☑ Center cutting

Tool down step: 0

Angle: 5

Radius: 5

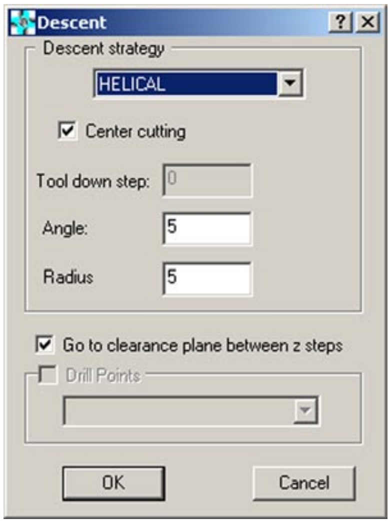
☑ Go to clearance plane between z steps

☐ Drill Points

If you select the **Angle** approach type and enter an **Angle** value, the tool will enter the material with a ramp movement. The ramp will start from the given safety distance down to the model contour. If, due to the model contour, the ramp cannot be created in a straight line, the approach movement will follow the contour.

When you work with small descent angles, e.g. 2 degrees, you could shorten the length of the descent movement by decreasing the safety distance value. Make sure, though, that the initial offset on the raw material allows for changing the safety distance value.

Helical



Descent

Descent strategy: HELICAL

☑ Center cutting

Tool down step: 0

Angle: 5

Radius 5

☑ Go to clearance plane between z steps

☐ Drill Points

Tool with Center cutting capabilities

If your cutting tool has Center cutting capabilities, select **Center cutting**. In the **Angle** field, enter the descent angle that you would like the tool to follow. In the **Radius** field enter the radius of the tool path.

The working order will be as follows:

- The **Helical Descent** will descend from the safety distance above the **Job upper plane** to the material in a circular motion until the down step is reached using the user-defined radius.
- When the tool reaches the down step depth, it will then machine all the material at the down step depth.
- When the machining is completed at this depth, it will then go up to the clearance plane.
- At this stage it will reposition itself at the **Z** safety distance above the previous down step depth and repeat the helical descent to the next working depth.
- This process will repeat itself until the final depth has been machined.

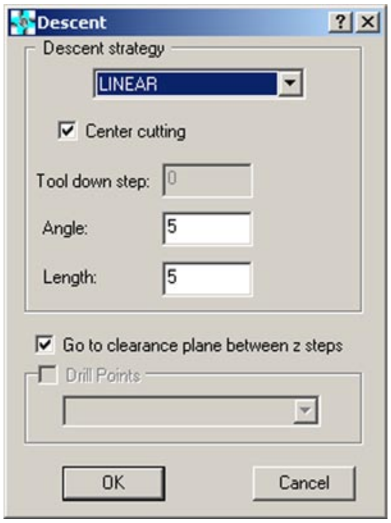
Tool without Center cutting capabilities

If the cutting tool does not have Center cutting capabilities, leave the **Center cutting** field empty. In the **Tool down step** field, enter the depth of the down step of the tool.

The result of this action is the same as with a center cutting tool except for the following differences:

- The tool descends from the safety distance above the Job upper plane to the material in a circular motion up to the Tool Down Step.
- At this point the tool will clear a circular path. It will then descend to the next Tool Down Step.
- This will be repeated until the tool reaches the CAM-Parts down step depth.
- When the tool reaches the down step depth, it will then machine all the material on that down step depth.
- When the machining is completed at this depth, it will then go up to the clearance plane.
- At this stage it will reposition itself at the Z safety distance above the previous down step depth and repeat the helical descent to the next working depth.
- This process will repeat itself until the final depth has been machined.

Linear



Descent

Descent strategy: LINEAR

☑ Center cutting

Tool down step: 0

Angle: 5

Length: 5

☑ Go to clearance plane between z steps

☐ Drill Points

The **Linear** descent follows the same rules as the **Helical**. The difference is that the descent will be in a linear zigzag fashion rather than a circular one.

Tool with Center cutting capabilities

If your cutting tool has Center cutting capabilities, select **Center cutting**. In the **Linear** dialog window enter the length of the linear tool path that you would like the tool to travel.

The working order will be as follows:

- The **Linear Descent** will descend from the safety distance above the Job upper plane to the material in a zigzag motion based on the user defined angle and length until the down step is reached.
- When the tool reaches the down step depth, it will then machine all the material on that down step depth.
- When the machining is completed at this depth, it will then go up to the clearance plane.
- At this stage it will reposition itself at the Z safety distance above the previous down step depth and repeat the linear descent to the next working depth.
- This process will repeat itself until the final depth has been machined.

Tool without Center cutting capabilities

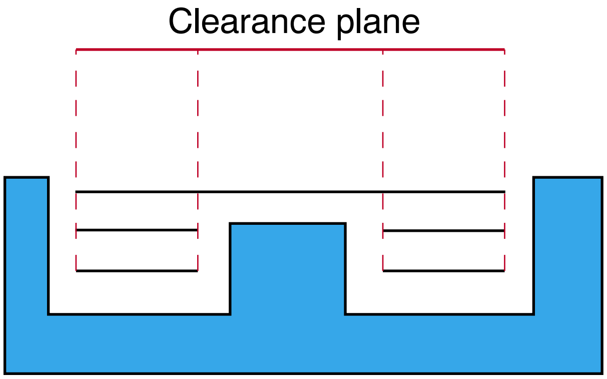
If the cutting tool does not have Center cutting capabilities, leave the **Center cutting** field empty. In the **Tool down step** field, enter the depth of the down step you want the tool go to.

The result of this action is the same as with a center cutting tool except for the following differences:

- The tool descends from the safety distance above the Job upper plane to the material in a zigzag motion up to the Tool Down Step.
- At this point the tool will clear a straight linear path. It will then descend to the next Tool Down Step.
- This will be repeated until the tool reaches the CAM-Part down step depth.
- When the tool reaches the down step depth, it will then machine all the material on that down step depth.
- When the machining is completed at this depth, it will then go up to the clearance plane.
- At this stage it will reposition itself at the Z safety distance above the previous down step depth and repeat the linear descent to the next working depth.
- This process will repeat itself until the final depth has been machined.

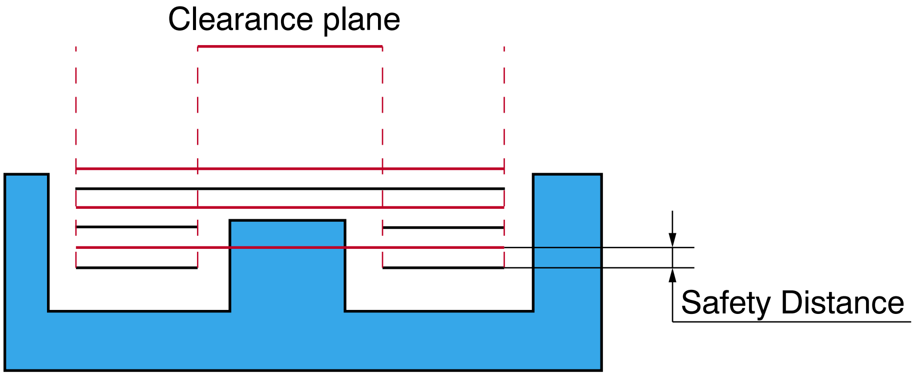
Go to clearance plane between Z steps

This option enables you to control the height, where the horizontal rapid movement will be performed.



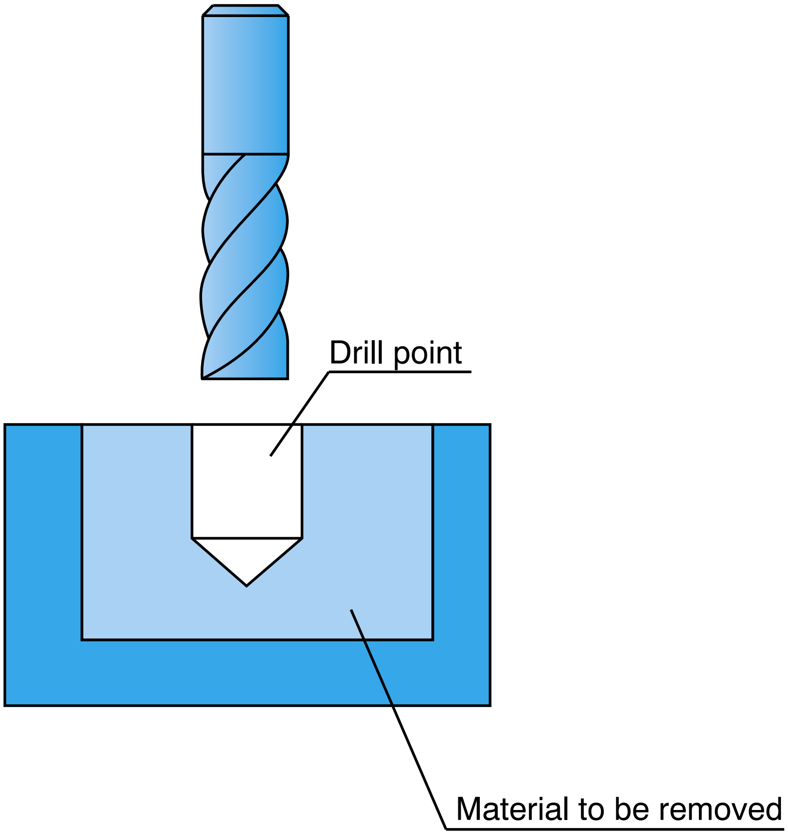
When the option is turned on (default state), SolidCAM performs the horizontal rapid movements on the **Clearance plane** of the Job.

When the option is turned off, SolidCAM performs the horizontal rapid movements on the safety distance from the last cut. SolidCAM checks for possible collisions. In areas where the rapid movements on the safety distance causes gouges, SolidCAM performs the horizontal rapid movements on the **Clearance plane**.



Drill points

SolidCAM enables you to perform the approach to the material from the hole, previously machined by the **3D Drill Job** (see the topic **7.7**.). The **3D Drill Job** has to be situated before the 3D Model Job in the CAM Manager. SolidCAM enables you to choose the geometry used for the 3D Drill Job in order to define the approach point.

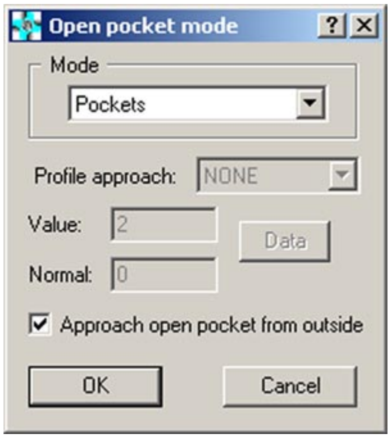


7.4.7.Mode (Open Pocket machining)

SolidCAM locates open pocket areas and enables you to machine them with a specific strategy. This strategy enables you to combine pocket and profile tool paths in order to provide the best productivity in open pocket machining.

The **Open Pocket machining** option is available to use within **3D Model Roughing** and **Constant-Z Floor machining strategy** (see the topic **7.5.8.**).

The **Mode** button displays the **Open pocket mode** dialog.



Mode: Pockets

Profile approach: NONE

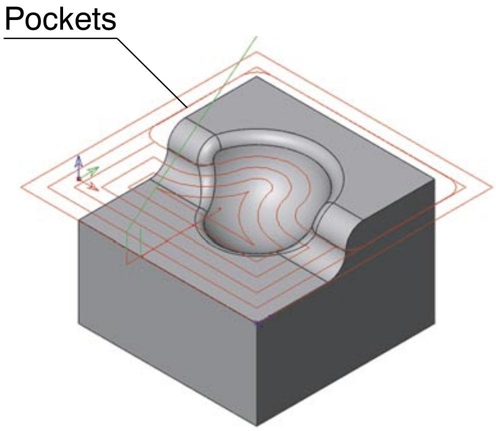
Value: 2

Normal: 0

☑ Approach open pocket from outside

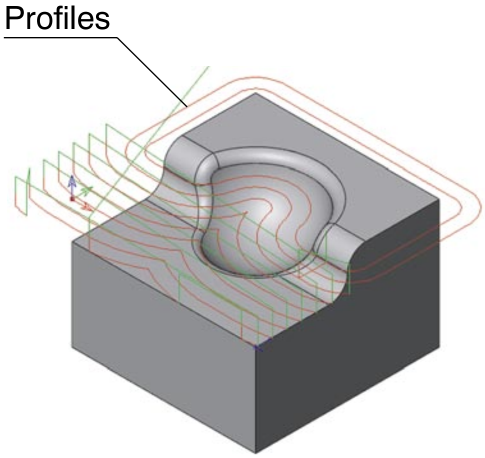
There are three modes for open pocket machining:

Pockets



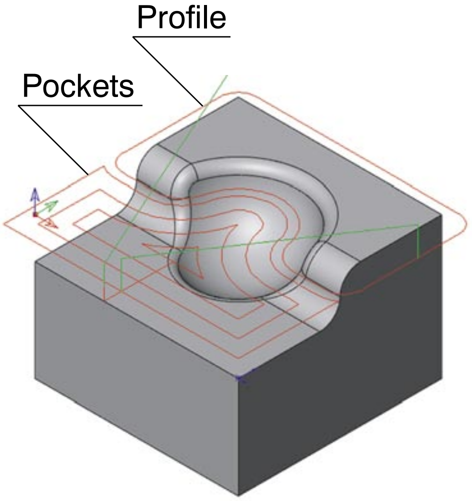
Both closed and open areas at each down step are milled as **Pockets**.

Profiles



- **Closed** areas at each down step are milled as **Pockets**.
- **Open** areas at each down step are milled by a number of equidistant **Profiles** from outside the model; the tool moves in parallel offsets to the contour of the section on the specified Z-Levels.

Pockets+Profile



- **Closed** areas at each down step are milled as **Pockets**.
- If the **Open** areas at each down step cannot be milled completely with one **Profile**, then those unmilled areas are milled first by **Pocket** paths and then the **Profile** is used for finish.

- The advantage of **Pockets+Profile** mode over **Pockets** is that in narrow places it performs a **Profile** toolpath instead of a **Pocket** toolpath thus saving time.
- The **Profiles** mode is useful when you are milling a boss-type model (e.g. Core of Mold).

Profile Approach

In the **Profile approach** combo-box, you can select one of the following options:

- **None**
- **Normal**
- **Arc**
- **Tangent**

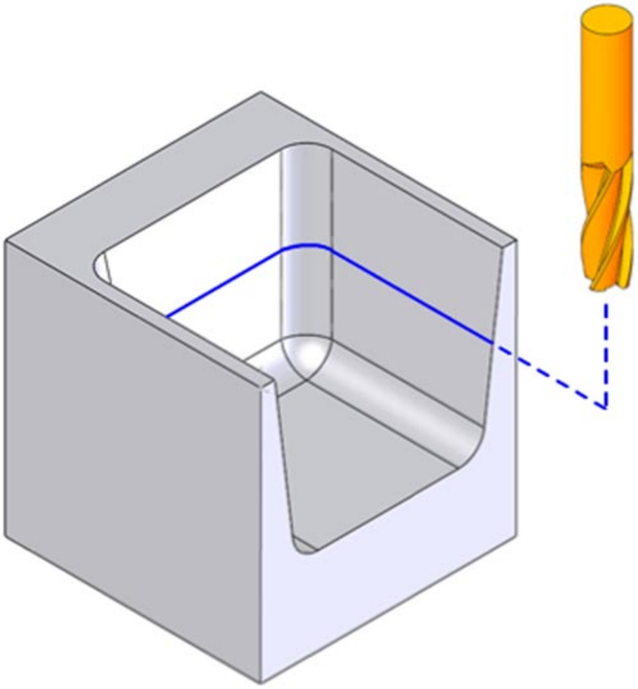
The **Profile approach** options are the same as **Approach** options in the **Profile Job**.

For more information, please refer to the topic **6.1.12**.

Approach open pocket from outside

The **Approach open pocket from outside** option enables the tool to approach from outside of the material in the open pocket areas, if possible. Such an approach enables you to decrease the tool loading when plunging into the material.

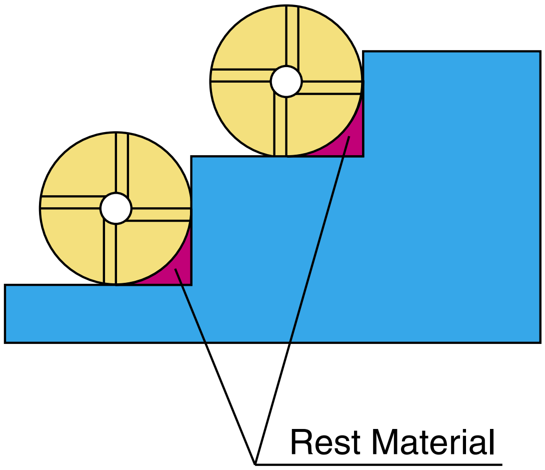
This option enables SolidCAM to perform the approach movement from the automatically calculated point outside of the material. The tool approaches to the necessary depth outside of the material and then plunges into the material.



In closed areas, where outside approach is impossible, the tool plunges into the material with a strategy defined in the **Descent** field of the Job screen.

7.4.8 Rest Material

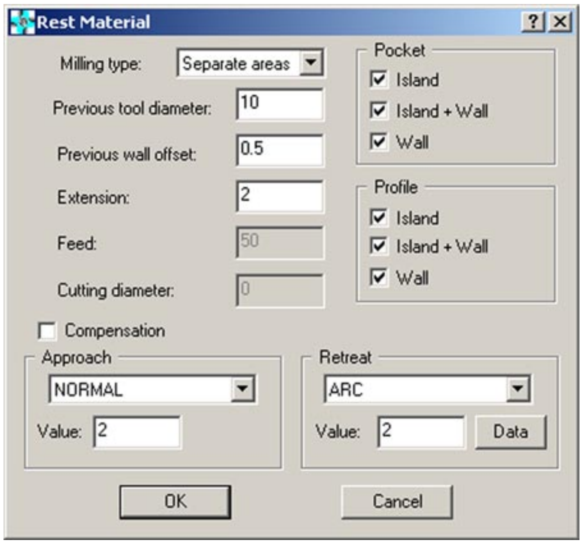
With the **3D Roughing strategy**, SolidCAM generates a number of sections parallel to the XY-plane of the CAM-Part coordinate system. These sections are generated with a Z-step defined either by constant down-step or by scallop. In each of these sections SolidCAM uses the section profile geometry to produce a pocket tool path. The **Rest material** feature enables SolidCAM to machine in each of these sections only in areas where the previous tool was not able to machine.



In **Profile Machining**, when a large tool is used around the profile, the tool will leave material in corners that it cannot enter.

The rest material option enables you to remove the material from this area without defining a new geometry.

In the **Rest Material dialog** box, the following data must be given:



Milling type: Separate areas
Previous tool diameter: 10
Previous wall offset: 0.5
Extension: 2
Feed: 50
Cutting diameter: 0

Pocket
☑ Island
☑ Island + Wall
☑ Wall

Profile
☑ Island
☑ Island + Wall
☑ Wall

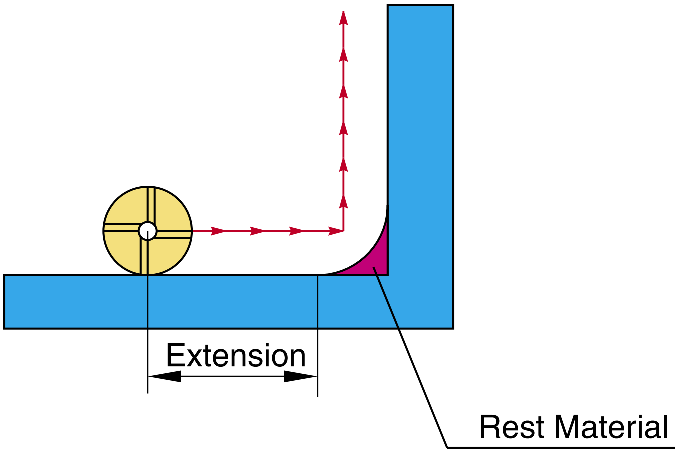
☐ Compensation

Approach: NORMAL
Value: 2

Retreat: ARC
Value: 2 Data

In this dialog you have to define the following parameters:

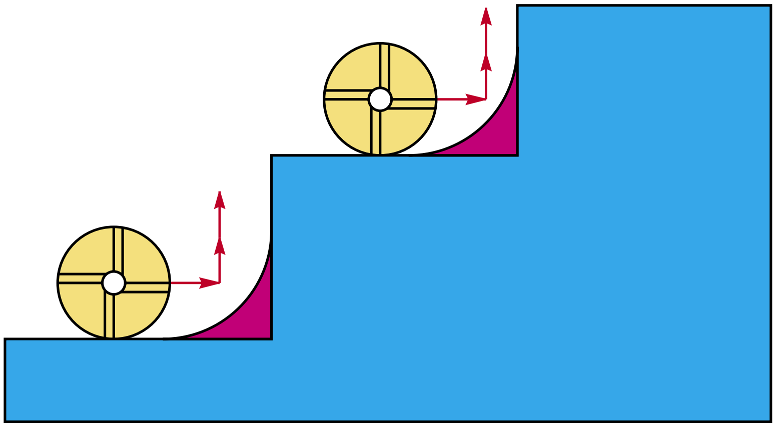
- **Previous tool diameter** - The diameter of the rough end-mill that was used in the previous job.
- **Previous wall offset** - The wall offset of the previous job.
- **Extension** - The overlap distance that you would like to start and end from the previous larger end-mill.



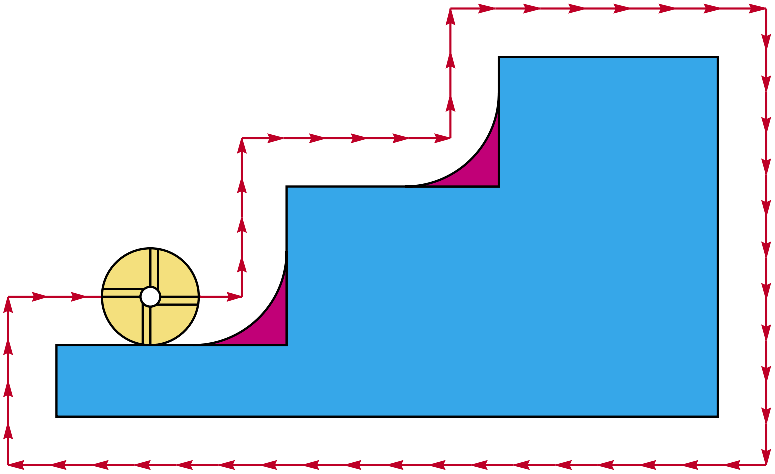
Milling type

Choose the milling type either as **Separate areas** or **Around profile**. In separate areas, SolidCAM generates a profile or pocket tool path to clean areas that the previous tool could not mill. The around profile strategy generates a closed profile to mill the rest material.

Separate areas - with this option SolidCAM machines only areas not machined with the previous tool.

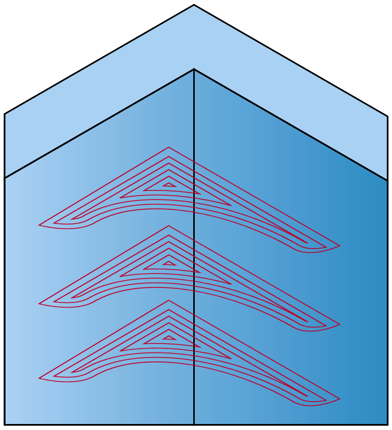


Around Profile - SolidCAM machines the whole profile.

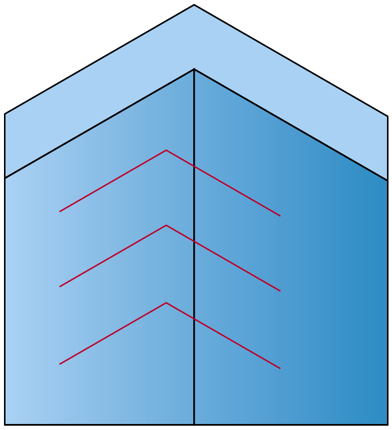


In **3D Model** and **Pocket Jobs** with **Separate Area** machining strategy, SolidCAM can generate two types of tool paths, depending on the volume of the rest material and on the previous and present tool diameters:

- **Pocket-style tool paths.**
- **Profile-style tool paths.**

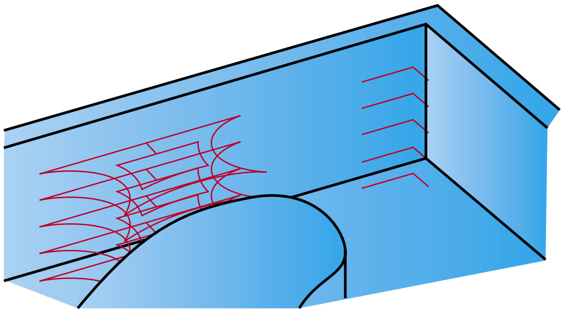


Pocket-style tool paths



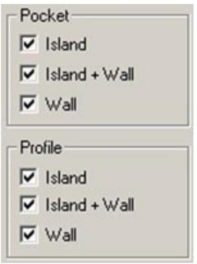
Profile-style tool paths

SolidCAM generates either a pocket or profile tool path in each area of the rest material.



The options picked in this box will determine which of these tool paths will be cut:

- **Island** – only tool paths that touch the Islands are cut.
- **Island + Wall** – tool paths that touch both the Wall and Islands are cut.
- **Wall** - only tool paths that touch the Wall are cut.



Pocket
☑ Island
☑ Island + Wall
☑ Wall

Profile
☑ Island
☑ Island + Wall
☑ Wall

Compensation

When the field Compensation is activated, SolidCAM will use tool radius compensation commands (G41, G42 for Fanuc) in G-Code output.

Approach

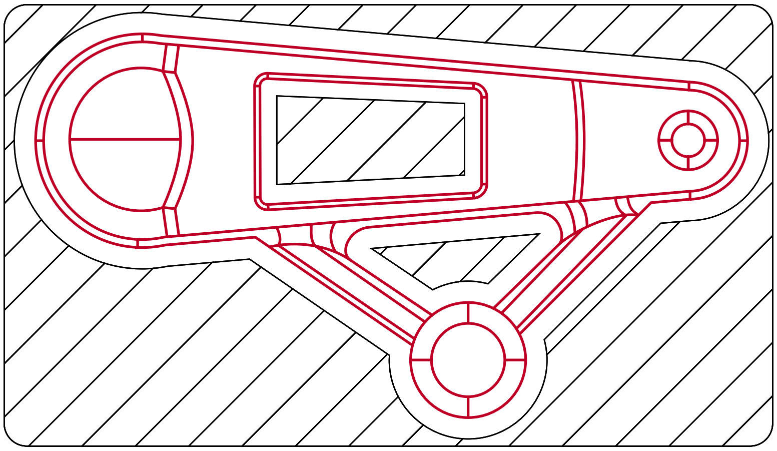
The approach and retreat on each Z-level can be controlled using the following options:

- **None**
- **Normal**
- **Arc**

For more information on these options, please refer to the topic **6.1.12**.

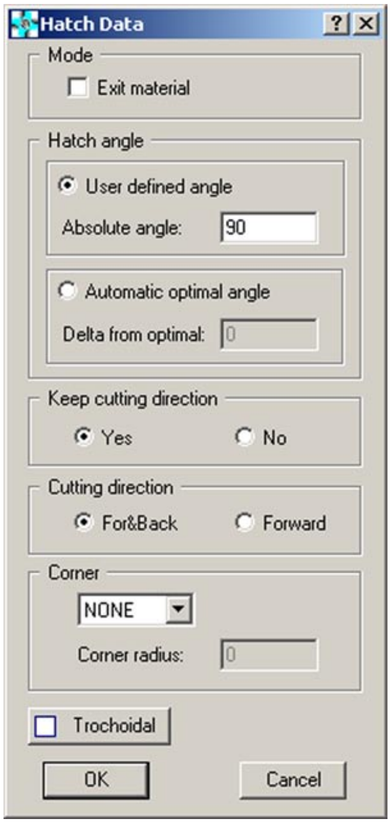
7.4.9 Hatch Roughing

The model will be roughed in a linear pattern. **Hatch rough** will create a linear raster tool path up to the model contour.



Click on the **Data** button. The **Hatch Data dialog** will be displayed.

For more information on the **Hatch** parameters, please refer to the topic **6.2.4**.



Hatch Data

Mode

☐ Exit material

Hatch angle

● User defined angle

Absolute angle: 90

○ Automatic optimal angle

Delta from optimal: 0

Keep cutting direction

● Yes ○ No

Cutting direction

● For&Back ○ Forward

Corner

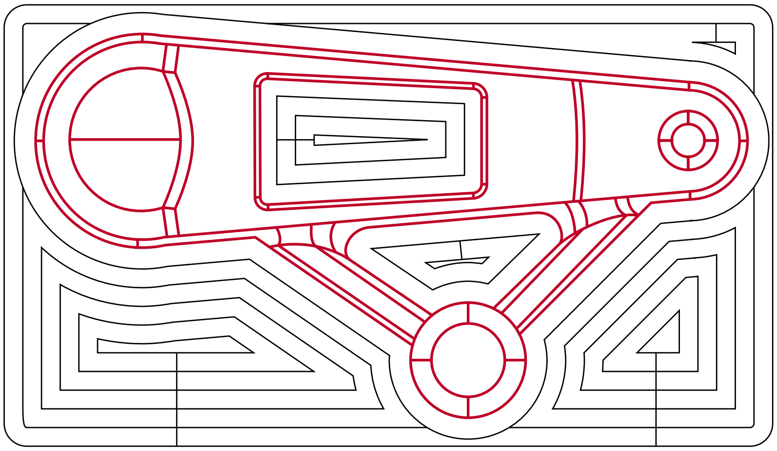
NONE

Corner radius: 0

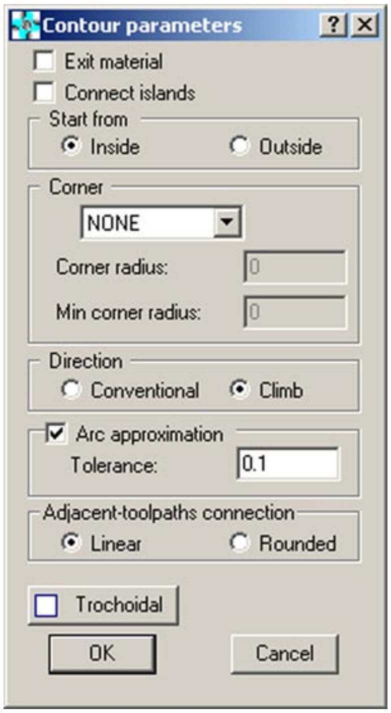
☐ Trochoidal

7.4.10 Contour Roughing

The **Contour rough** strategy enables SolidCAM to produce Constant-Z pockets on different Z-levels. Within the pockets, the tool moves in parallel offsets to the contour of the section of the model on each down step.



Click on the **Data** button. The **Contour Data dialog** will be displayed.



Contour parameters

☐ Exit material

☐ Connect islands

Start from: ● Inside ○ Outside

Corner: NONE

Corner radius: 0

Min corner radius: 0

Direction: ○ Conventional ● Climb

☑ Arc approximation

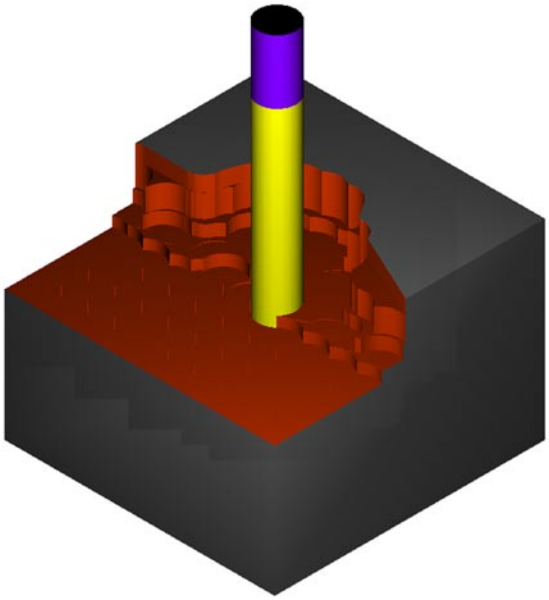
Tolerance: 0.1

Adjacent-toolpaths connection: ● Linear ○ Rounded

☐ Trochoidal

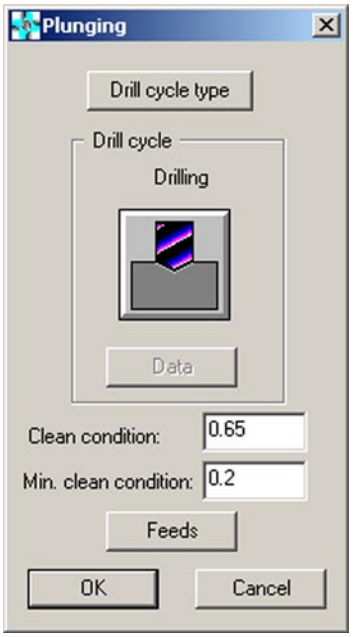
For more information on the **Contour** parameters, please refer to the topic **6.2.5**.

7.4.11 Plunging pattern



Plunging is a totally different concept of removing material from a given pocket with a special tool. Instead of milling the material, the tool moves up and down as in a drilling motion, traveling along the path type you choose.

Click on the **Data** button. The **Plunging dialog** will be displayed.



Plunging
Drill cycle type
Drill cycle
Drilling
Data
Feeds
OK
Cancel

Clean condition: 0.65

Min. clean condition: 0.2

For more information on the **Plunging** parameters, please refer to the topic **6.2.8**.

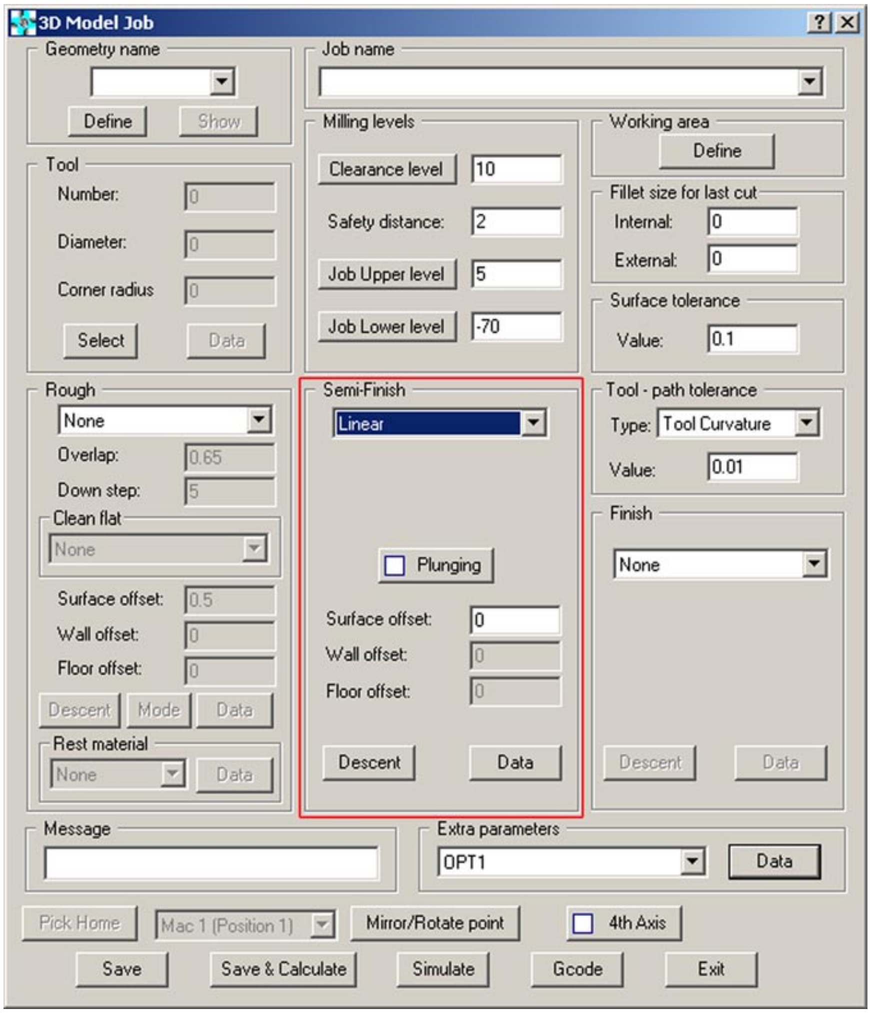
7.5 Semi-Finish/Finish

7.5.1 Overview

Semi-Finish machining prepares the model for finishing. After semi-finishing the model, a uniform offset will remain on the material. This offset will be removed in a last finishing cut. In contrast to rough operations, in conventional semi-finishing operations higher feed rates and narrower tolerances are applied.

The same machining strategies in Finish machining can be applied for Semi-finishing. It only differs in one additional parameter, Surface Offset, that controls the X,Y and Z offset that remains on the original surface. Otherwise all strategies are defined and used in the same way as in finishing.

In the **Semi-Finish** area, select a **strategy** from a list.


3D Model Job

Tool
Number: 0
Diameter: 0
Corner radius: 0

Milling levels
Clearance level: 10
Safety distance: 2
Job Upper level: 5
Job Lower level: -70

Fillet size for last cut
Internal: 0
External: 0

Surface tolerance
Value: 0.1

Rough: None
Overlap: 0.65
Down step: 5
Clean flat: None
Surface offset: 0.5
Wall offset: 0
Floor offset: 0
Rest material: None

Semi-Finish: Linear
☐ Plunging
Surface offset: 0
Wall offset: 0
Floor offset: 0

Tool - path tolerance
Type: Tool Curvature
Value: 0.01

Finish: None

Message

Extra parameters: OPT1

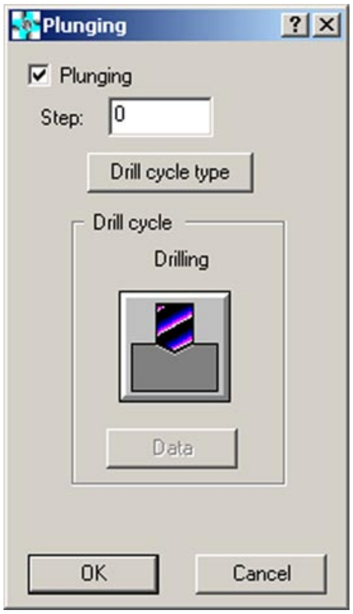
Mac 1 (Position 1)
☐ 4th Axis

Geometry name
Define
Show
Job name
Working area
Define
Select
Data
Descent
Mode
Data
Data
Descent
Data
Descent
Data
Data
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

Plunging

SolidCAM enables you to use the special plunging strategies for semi-finishing.

1. Click on the **Plunging** button.
2. Set the plunging parameters in the **Plunging dialog**.

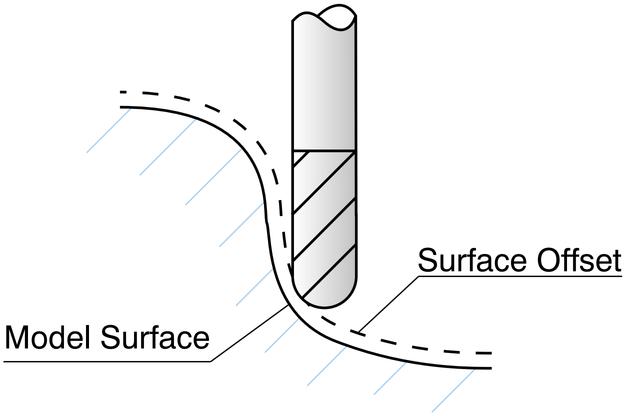

Plunging
☑ Plunging
Step: 0
Drill cycle type
Drill cycle
Drilling
Data
OK
Cancel

For more information on the **Plunging** parameters, please refer to the topic **6.2.8**.

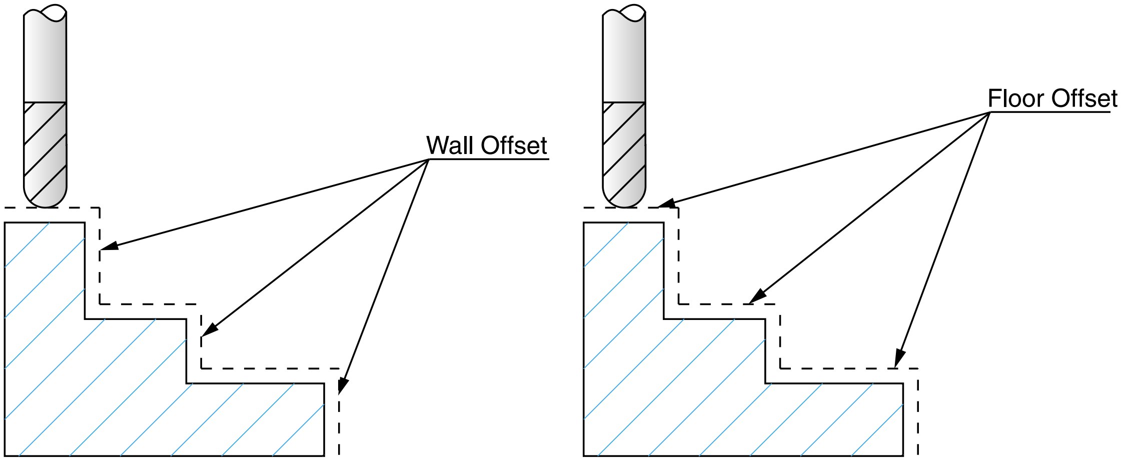
Offsets

- **Surface offset** - a minimum offset of material in the X, Y and Z direction will remain on the model contour. The value defines the closest distance the tool can get to the model.

In addition to the specified **Surface offset**, the roughed model will show stairs of rest material with a height equal to the down step value. In order to obtain a uniform offset on the model, a semi-finish operation should be executed.


Surface Offset
Model Surface

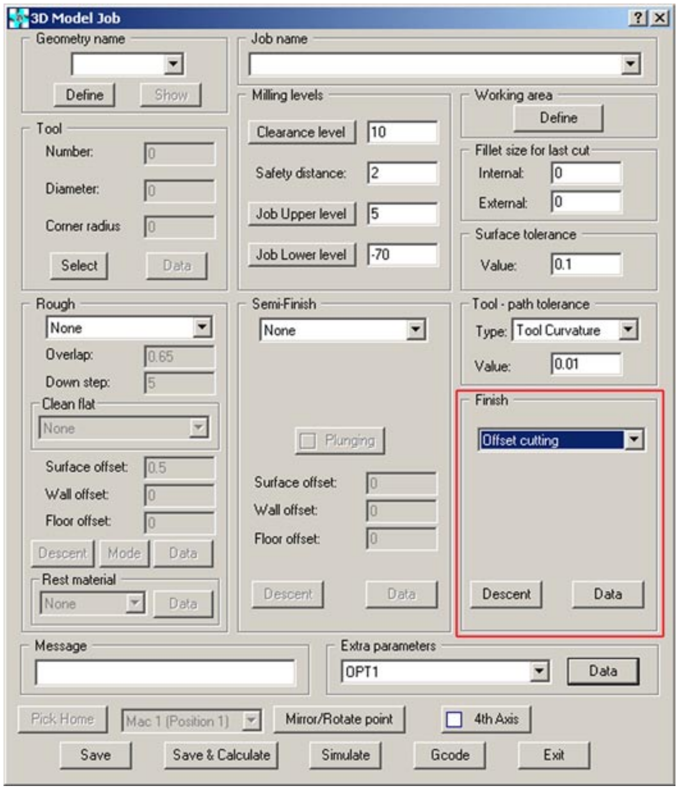
- In 3D milling of prismatic parts we often need to use the different offset values for vertical and horizontal areas. With the **Wall Offset** value you can define the XY offset of any surface that is not absolutely flat. To define the Z offset to the absolute horizontal (flat) surfaces, use the **Floor Offset** parameter.


Wall Offset
Floor Offset

- Use the **Wall** and **Flat** offsets available to set different offset values to the floor and walls only in **Constant-Z machining**.
- If you would like to use negative offsets, e.g. for plunge EDM electrode milling, the corner radius of the tool must be equal or larger than the specified offset value.

Finish machining generates the tool path directly on the surfaces and solids of your 3D model geometry. No offset can be specified and the model will be machined to its final topology and dimensions. You can apply a number of different machining strategies to finish your models. Each one of the strategies offers parameters to adjust the strategy to your needs.

The 3D job dialog offers all three machining types, namely: Rough, Semi-Finish and Finish Milling in one screen. This enables you to perform all machining operations in one job. You will need to add another job only if the tool, milling levels or the working area is going to change from one operation to the other.


3D Model Job
Geometry name
Define
Show
Job name
Milling levels
Working area
Define
Tool
Select
Data
Fillet size for last cut
Surface tolerance
Rough
Semi-Finish
Tool - path tolerance
Finish
Descent
Mode
Data
Descent
Data
Descent
Data
Message
Extra parameters
Data
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

Tool: Number: 0; Diameter: 0; Corner radius: 0

Clearance level: 10
Safety distance: 2
Job Upper level: 5
Job Lower level: -70

Fillet size for last cut: Internal: 0; External: 0
Surface tolerance: Value: 0.1
Tool - path tolerance: Type: Tool Curvature; Value: 0.01

Rough: None
Overlap: 0.65
Down step: 5
Clean flat: None
Surface offset: 0.5
Wall offset: 0
Floor offset: 0
Rest material: None

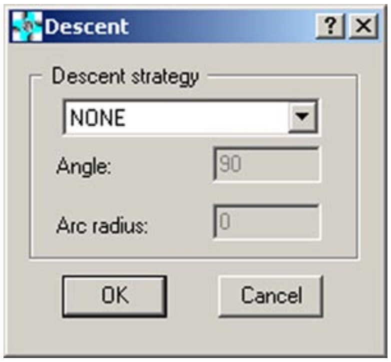
Semi-Finish: None
☐ Plunging
Surface offset: 0
Wall offset: 0
Floor offset: 0

Finish: Offset cutting

Extra parameters: OPT1
Mac 1 (Position 1)
☐ 4th Axis

7.5.2 Descent

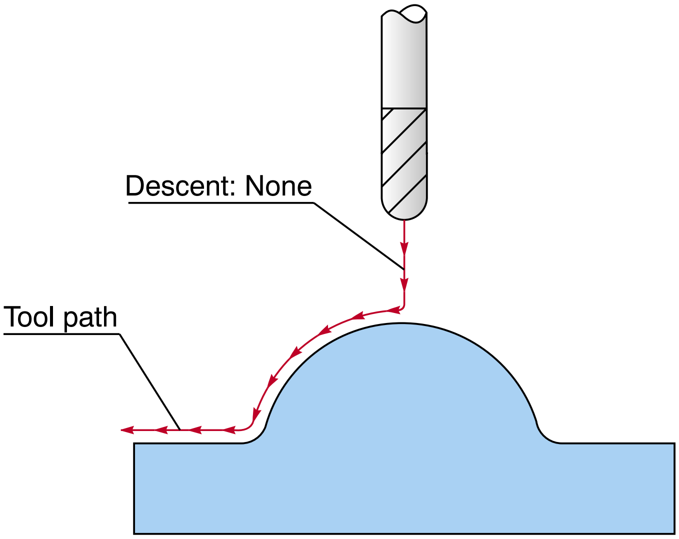
You can control how the tool will plunge into the material. Click on the **Descent** button to display the **Descent** dialog.


Descent
?
Descent strategy
OK
Cancel

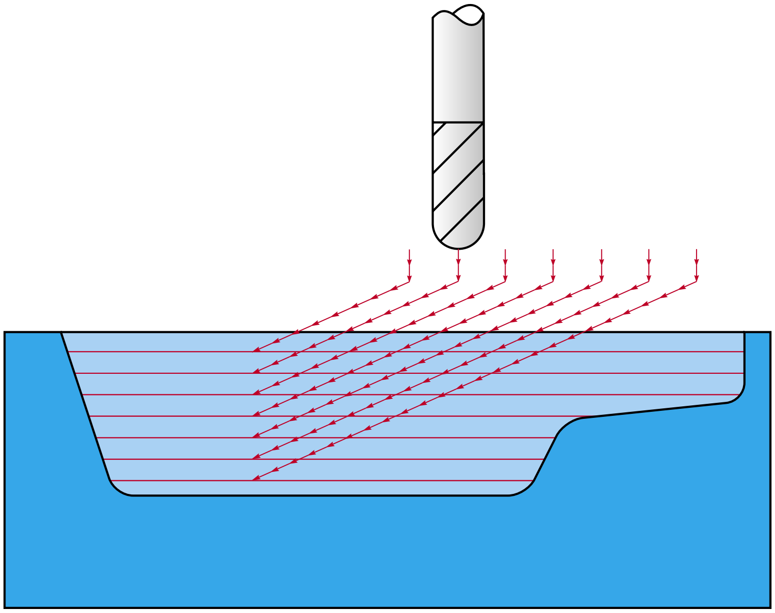
Descent strategy: NONE
Angle: 90
Arc radius: 0

- **None**

  When Approach is set to **None** or the default angle value is 90 degrees, the tool will plunge into the material with a normal downward movement at the specified Z-feed rate.

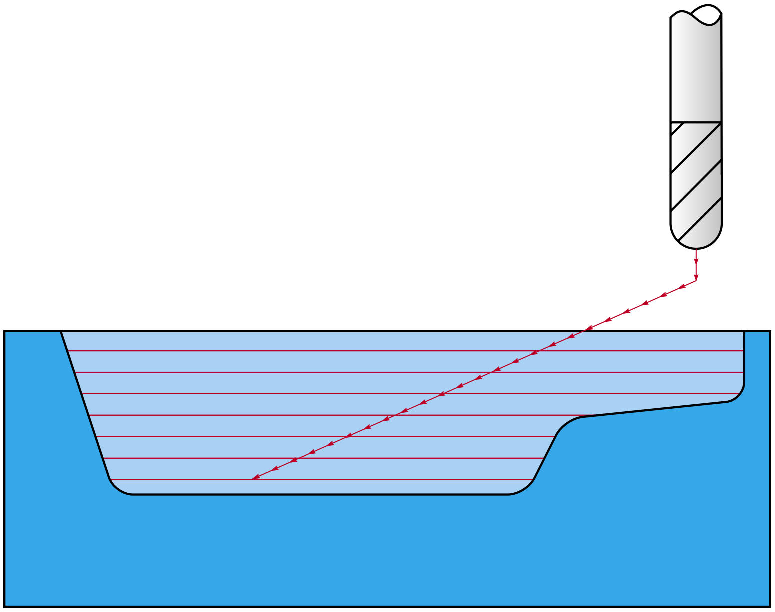

Descent: None
Tool path

- **Angle along path**



The tool will plunge with a ramp movement. The ramp will start from the given safety distance down to the model contour on the current Z-level. If, due to the model contour, the ramp cannot be created in a straight line, the approach movement will follow the contour.

- **Angle between paths**

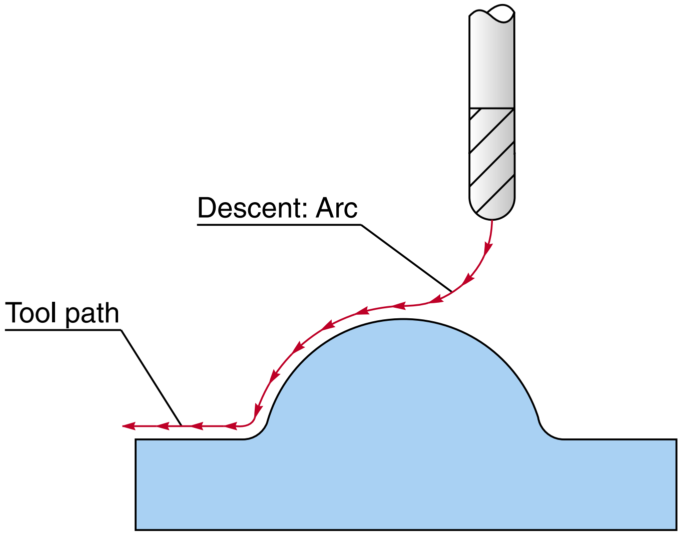


The tool will plunge with a ramp movement. The ramp movement for the first Z-level will start from the given safety distance. The next ramp movements will be performed between the Z-levels. If, due to the model contour, the ramp cannot be created in a straight line, the approach movement will follow the contour.

The **Along between paths** option is available only with Constant Z finish strategy.

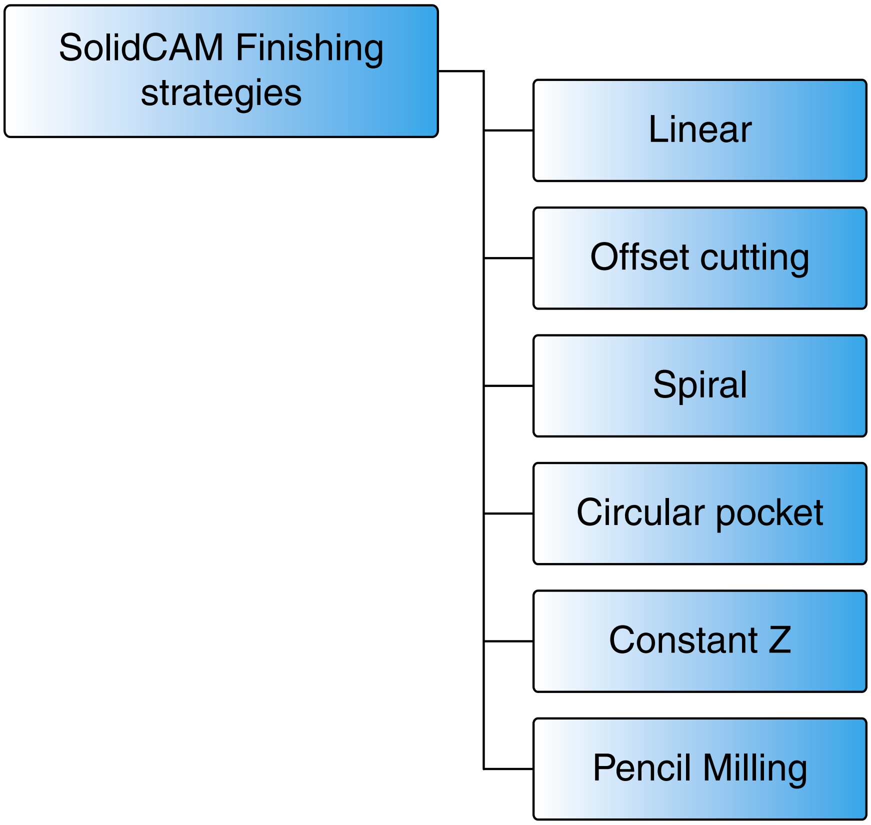
- **Arc**

The tool will plunge using an arc movement (with a specified radius) in the vertical plane.



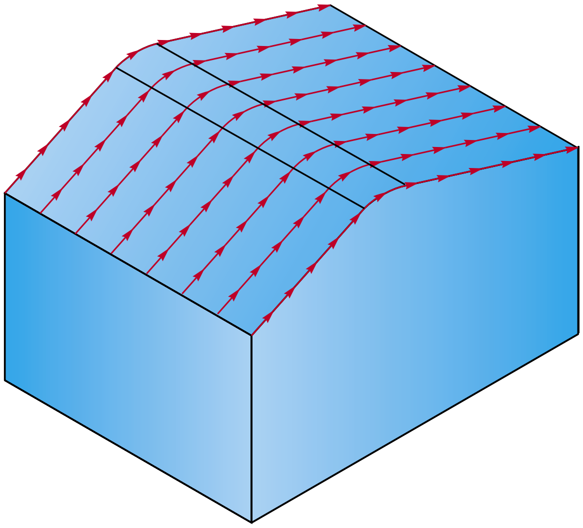
This option is not available with the **Constant-Z Finish** strategy.

7.5.3 Semi-Finish/Finish strategies

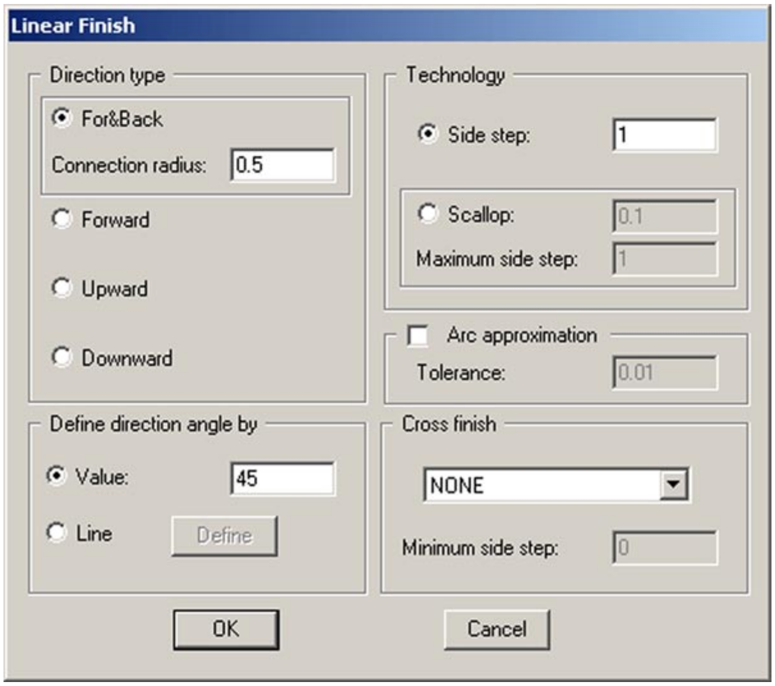

SolidCAM Finishing strategies
Linear
Offset cutting
Spiral
Circular pocket
Constant Z
Pencil Milling

7.5.4 Linear finish strategy

In linear cutting, the lines of a linear pattern are generated on a 2D plane above the model. The linear pattern is then projected on the 3D model. During this projection, the Z-position of the tool is calculated to avoid gouging of the material.



The **Linear Finish** dialog dialog displays parameters of the Linear Finish.



Linear Finish

Direction type
● For&Back
Connection radius: 0.5
○ Forward
○ Upward
○ Downward

Technology
● Side step: 1
○ Scallop: 0.1
Maximum side step: 1

☐ Arc approximation
Tolerance: 0.01

Define direction angle by
● Value: 45
○ Line

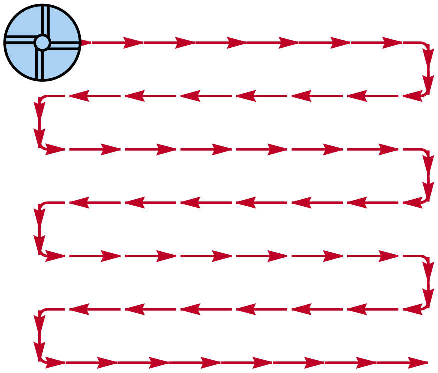
Cross finish
NONE
Minimum side step: 0

Direction type

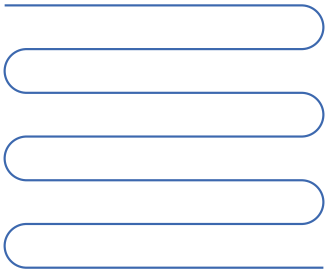
These options control the direction of the tool on each line of the linear pattern that has been projected on the model. Different materials and cutters will require specific cutting conditions, e.g. some tools will give better surface finish when cutting only downward - in other applications you will need to use other settings to achieve optimum results.

- **For&Back**

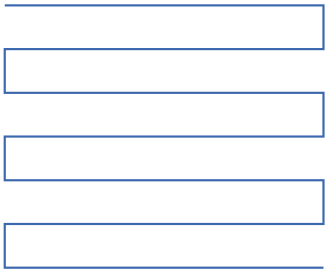
  The tool finishes one line of the linear pattern and then directly moves to the next line and so on. It mills forward and backward without leaving the material - thus constantly changing between climb milling and conventional milling.



Connection Radius - You can connect lines of the linear tool path with a radius. If the **Radius value** is **0**, SolidCAM connects the linear tool path with straight lines.



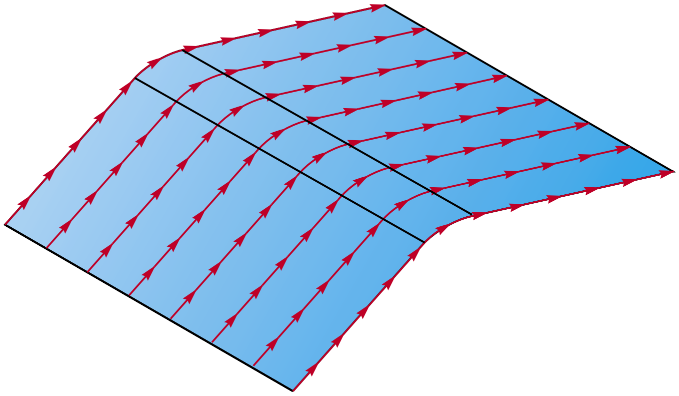
Connection with radius



Connection without radius

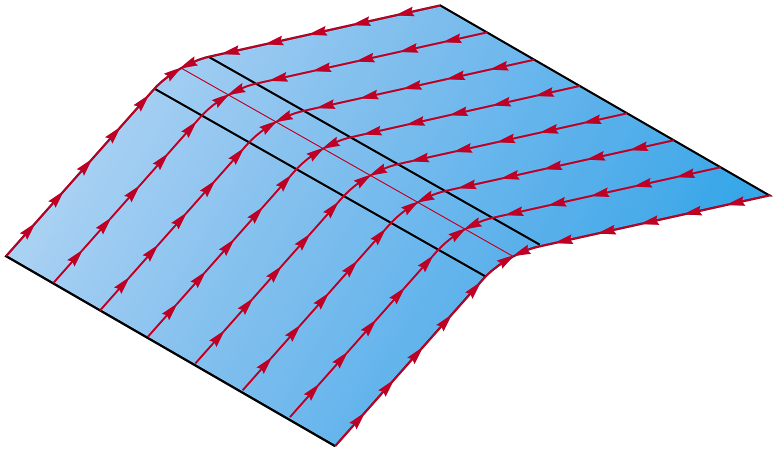
- **Forward**

  The tool finishes one line of the linear pattern. At the end of the line, the tool rapids (G0) to the safety distance and then to the start of the next cut line. The tool will always use climb or conventional milling.

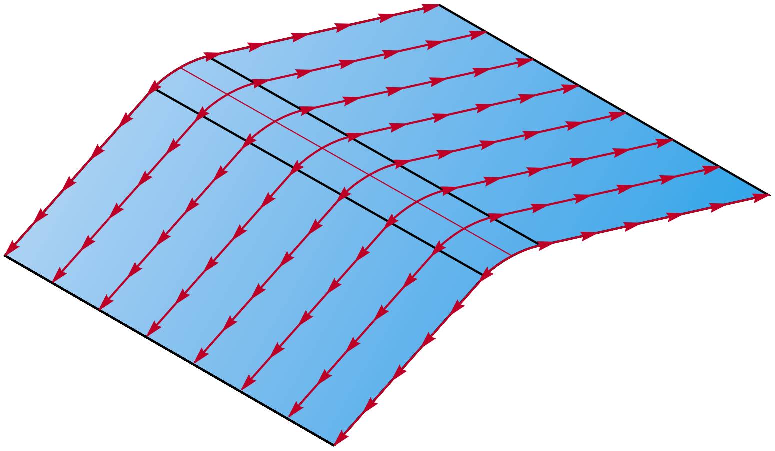


- **Upward**

The tool will always cut from the lowest point of the tool path line to the highest point. From the highest point of the cut, the tool will move rapidly to the start point of the next cut.



- **Downward**



The tool will always cut from the highest point of the tool path line to the lowest point. From the lowest point of the cut, the tool will then move rapidly to the start point of the next cut.

Define Direction

- **By Angle**

  Enter the angle that defines the orientation of the tool path. A value of 0 will generate the tool path lines in the direction of the positive X-axis.

- **By Line**

  When you choose the **Line** option, the **Define** button will be activated and you can define a line that determines the orientation of the tool path. The direction vector is defined by picking two points that define the start point and end point of the direction line. The angle of the selected line will be output to the angle field.

Technology

- **Side Step (constant)**

  In linear finishing, SolidCAM generates a line pattern on a 2D plane above the model and then projects it on the 3D model. The side step value determines the constant distance between all lines of the linear pattern created on the 2D plane before it is projected.

- **Scallop**

  This option enables you to control the distance between the tool path lines by the cusp height you want to achieve on the finished model. SolidCAM will adjust the distance between the single lines of the 2D pattern (before projection on the model) to the topology of the model to match the specified scallop height.

- **Maximum side step**

  On flat areas, the bull nose cutter could achieve a small scallop with a rather large side step. You can use this value if you want to limit the chip load on the tool or the cutting forces that a large side step could cause.

- **Arc approximation**

  You can create G2/G3 G-Code output from Linear finish operations. SolidCAM checks whether successive points of the calculated tool path can be connected using an arc or circle. If an arc or circle connection within the specified arc approximation tolerance can be made, you will receive arc and circle interpolation commands G2 and G3 in the generated G-Code. This feature can drastically reduce the number of lines in G-Code files. Most CNC-controllers and machines work much faster on arcs and circles than on single tool path points or splines. Arc approximation will increase actual feed rates on older CNC-machines and the machine will work smoother.

  The **Tolerance** value defines the tolerance SolidCAM uses to position tool path points on arcs or circles.

Cross finish

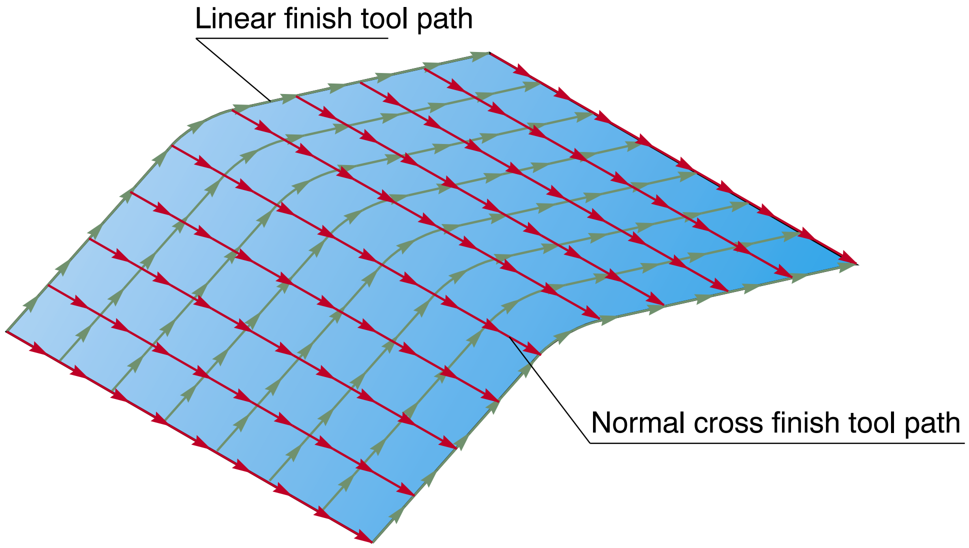
Cross finish type

- **None**

Linear machining will be performed only in the direction you have specified and no cross finish is executed.

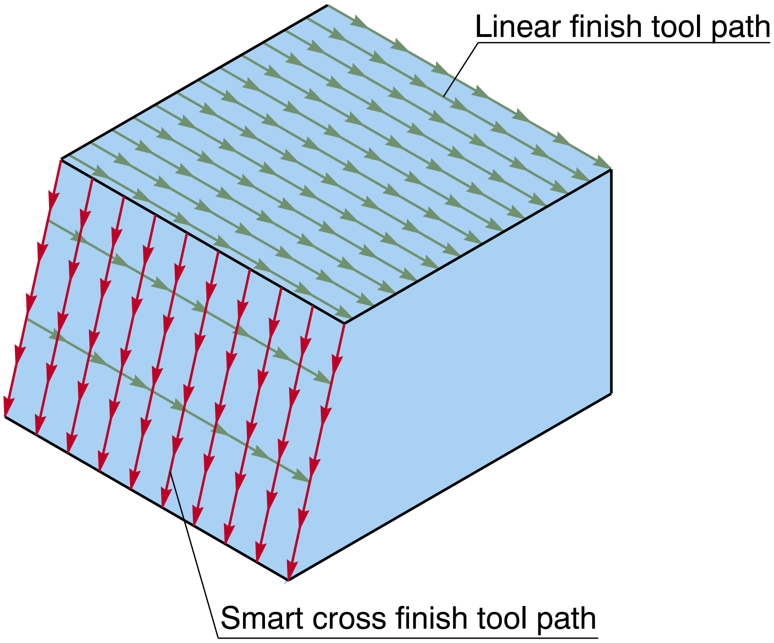
- **Normal**

After finishing the model in the first specified direction, another linear finish is performed at an angle of **90°** to the first cutting direction. The **Normal** cross finish option will re-machine the complete area that has been machined in the first cut linear operation


Linear finish tool path
Normal cross finish tool path

- **Smart**

The **Smart** cross finish option will only re-machine areas that have not been finished within the given scallop during the first linear finish operation at an angle of 90° to the first cut. The values set in the **Maximum side step** and the **Minimum side step** affect the calculation. If the scallop can be achieved using side steps larger than the minimum side step, no cross finish is necessary - some other constellation could result in all the working area to be re-machined.



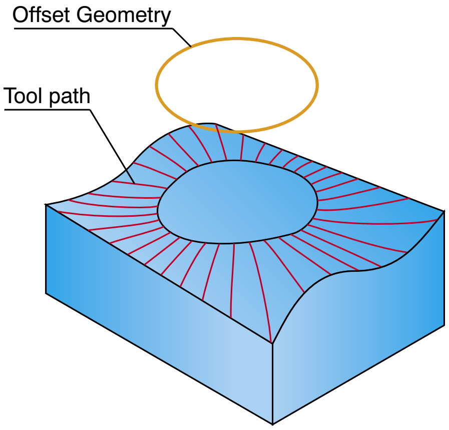
Minimum side step

This value can be used to avoid tool paths that are too narrow. You can, for example, enter a scallop of 0.02. For some areas SolidCAM would calculate a distance between two lines with 0.01 or less (at very steep walls). To reduce machining time and to avoid long G-Code files, you could then limit the minimum side step of the tool to 0.05.

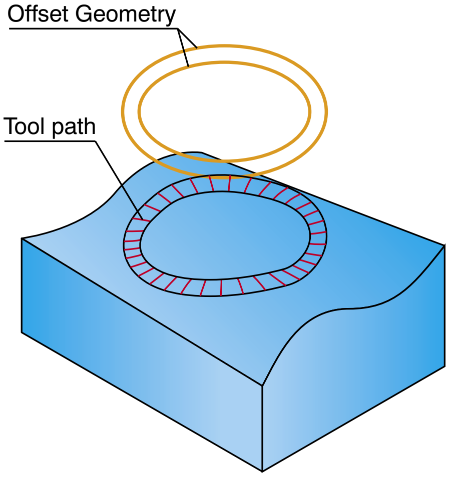
7.5.5 Offset cutting Finish Strategy

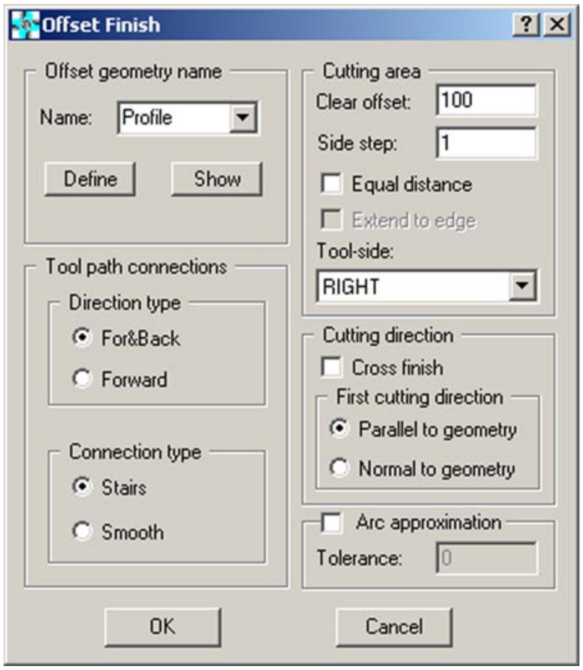
Offset cutting is a form of 3D profile milling that can be used to mill specific areas of the model. This strategy offers two possible alternatives of profile selection:

- If you select one Profile, you must specify the clear offset distance. On the right side of the profile, relative to the chain direction, the machining will be performed until the specified clear offset distance has been reached. In this example, the **Offset Profile** has been machined normal to the profile geometry.



- When you define two **Offset Profiles**, both profile geometries are connected and the tool path is projected on the 3D model.




Offset Finish
Offset geometry name
Name: Profile

Tool path connections
Direction type
● For&Back
○ Forward
Connection type
● Stairs
○ Smooth
Cutting area
Clear offset: 100
Side step: 1
☐ Equal distance
☐ Extend to edge
Tool-side: RIGHT
Cutting direction
☐ Cross finish
First cutting direction
● Parallel to geometry
○ Normal to geometry
☐ Arc approximation
Tolerance: 0

The **Offset Finish** dialog displays parameters of the Offset Finish.

Offset geometry

In this section, specify which profile geometry you want to use for the offset finish.

Cutting area

- **Clear offset**

  This value determines the offset distance from the profile that will be machined in the right direction of the geometry. This field is active if you use an offset geometry that consists of only 1 profile. In case the geometry has 2 profiles, this field will be inactive, as the profiles will be automatically connected.

- **Side Step**

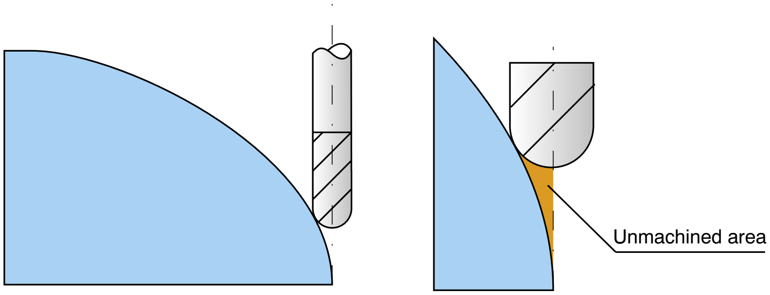
  In offset finish, SolidCAM generates the offset pattern on a 2D plane above the model and then projects it on the 3D model. The side step value determines the constant distance between all lines of the offset pattern created on the 2D plane before it will be projected.

- **Equal distance**

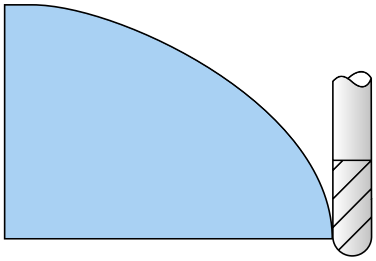
  By tagging **Equal distance** you can keep an equal distance between the tool paths.

- **Extend to edge**

  If this option is inactive, the tool path will be generated up to the position where the tool center is aligned with the end-point of the face; this could leave an unmachined area. When this option is active, the tool path is extended till the tool is tangent to the face, at the end-point of the face.



The **Extend to Edges** option is turned off

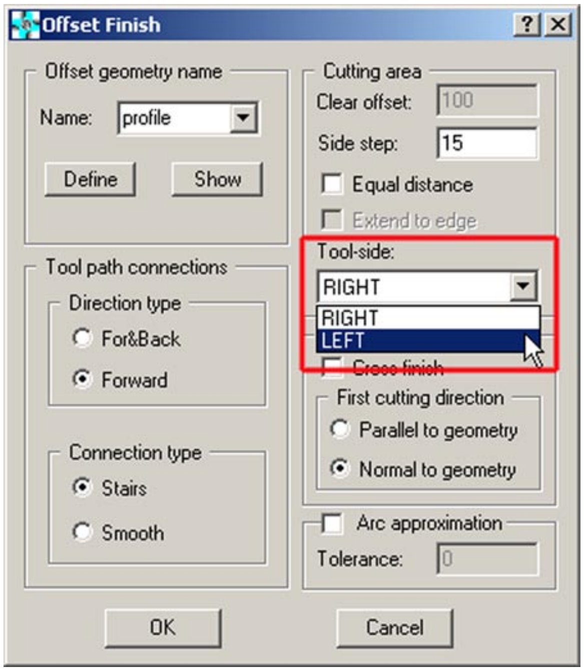


The **Extend to Edges** option is turned on

This option is available only if the **Equal Distance** option is activated.

- **Tool side**

This option enables you to define the working side of the geometry.



Offset Finish

Offset geometry name

Name: profile

Cutting area

Clear offset: 100

Side step: 15

☐ Equal distance

☐ Extend to edge

Tool-side: RIGHT

☐ Cross finish

Tool path connections

Direction type

○ For&Back

● Forward

Connection type

● Stairs

○ Smooth

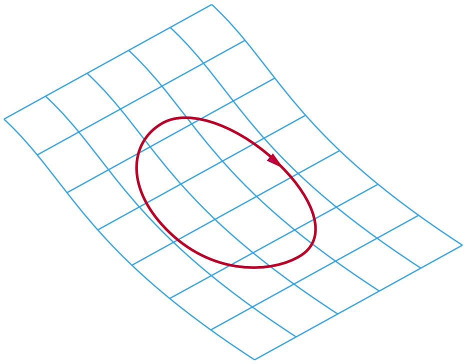
First cutting direction

○ Parallel to geometry

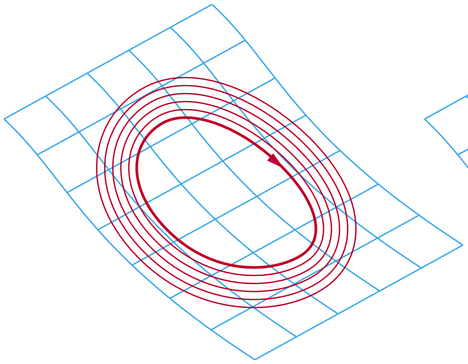
● Normal to geometry

☐ Arc approximation

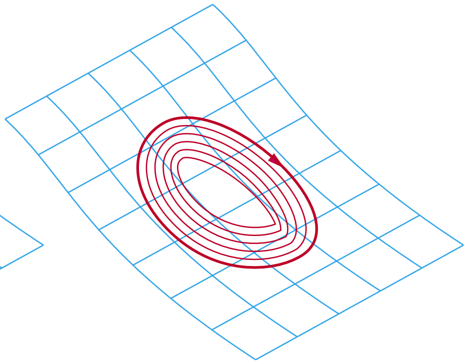
Tolerance: 0



Geometry orientation



Left side machining

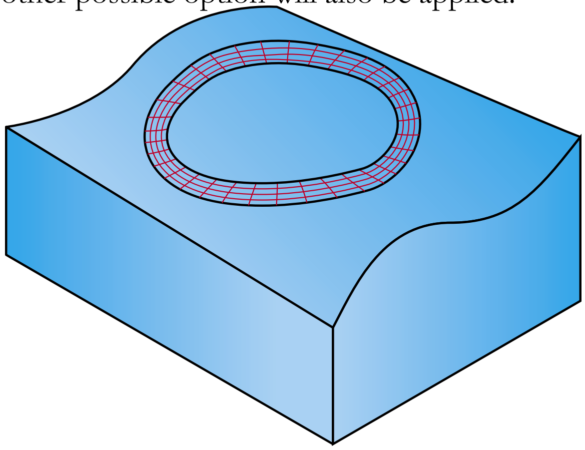


Right side machining

Cutting direction

Cross Finish

Check this field if you want to perform a **Cross finish** on the offset area. After finishing the model in the first specified direction, i.e. parallel or normal to the profile geometry, the other possible option will also be applied.



First cutting direction

If you have checked the cross finish field, both cutting directions will be applied to the offset area. Select which you want to use first. When no cross finish is used, only the selected option will be performed.

- **Parallel to geometry**

  SolidCAM will generate a pattern with offset lines parallel to the offset profile geometry. This pattern will then be projected on the model.

- **Normal to geometry**

  SolidCAM will generate a pattern with offset lines normal (90 degrees) to the offset profile geometry. This pattern will then be projected on the model.

Tool path connections

Direction type

- **For&Back**

The tool finishes one line of the offset pattern and then moves directly to the next line and so on. It mills forward and backward without leaving the material - thus constantly changing between climb milling and conventional milling.

- **Forward**

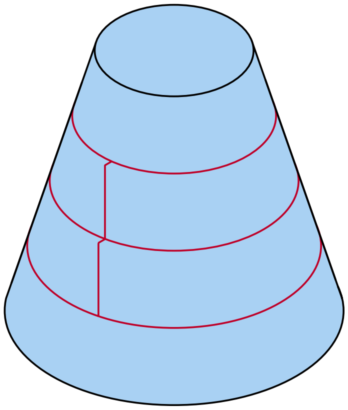
The tool moves finishes one line of the linear pattern. At the end of the line, the tool rapids (G0) to the safety distance and then to the start of the next cut line. The tool will always use climb or conventional milling.

Connection type

This option enables you to control the way the tool moves between two adjacent tool paths.

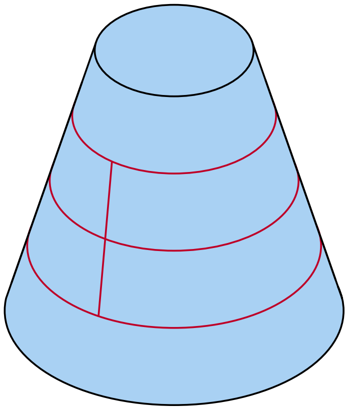
- **Stairs**

When the **Stairs** option is activated, the tool will move in two steps between two adjacent tool paths. If the next path is lower than the present path, the tool first moves away from the surface in the XY direction and then moves downwards in Z. In case the next path is higher than the present path, the tool will first move upwards in Z and then approach the surface in the XY direction.



- **Smooth**

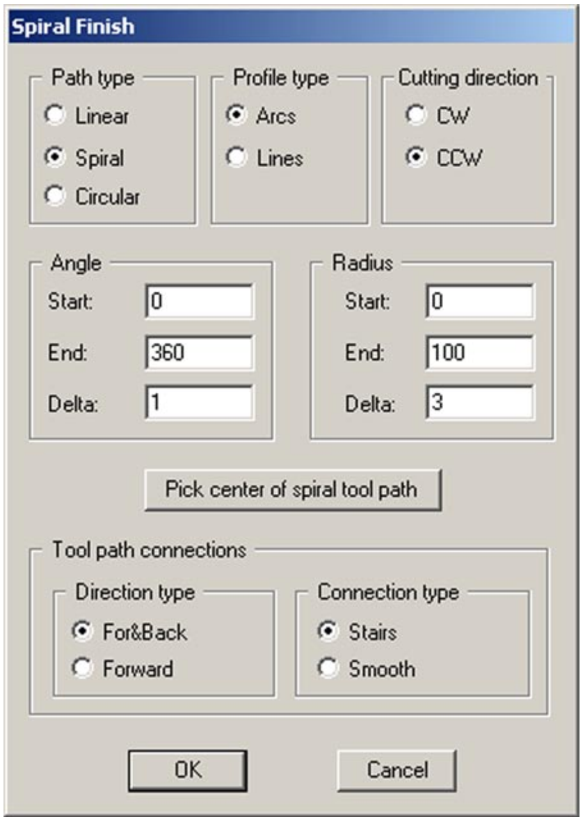
Between two adjacent tool paths, the tool moves on the surface of the model. The line connecting the two adjacent tool paths is projected on the model. When you use the **Smooth** option, the tool will not leave the material when moving to the start point of the next tool path.



7.5.6 Spiral Finish Strategy

Five different types of spiral tool path movements can be selected for this finishing strategy. The spiral tool path pattern is first generated in 2D and is then projected on the 3D model. During this projection, the Z-position of the tool is calculated so as to prevent gouging of the model.

The **Spiral Finish** dialog displays parameters of the Spiral Finish.



Path type
○ Linear
● Spiral
○ Circular

Profile type
● Arcs
○ Lines

Cutting direction
○ CW
● CCW

Angle
Start: 0
End: 360
Delta: 1

Radius
Start: 0
End: 100
Delta: 3

Tool path connections

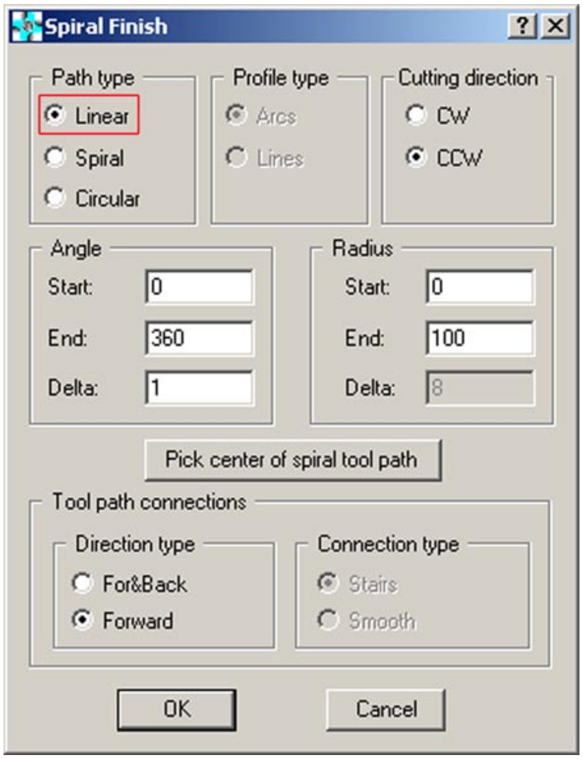
Direction type
● For&Back
○ Forward

Connection type
● Stairs
○ Smooth

Tool path type

By combining the options in the **Path type** and **Profile type** fields, you can generate 5 different kinds of tool path spirals. For each of the combinations you can set additional parameters in the **Angle** and **Radius** sections that will affect the spiral.

Path type: Linear



Spiral Finish

Path type
● Linear
○ Spiral
○ Circular

Profile type
● Arcs
○ Lines

Cutting direction
○ CW
● CCW

Angle
Start: 0
End: 360
Delta: 1

Radius
Start: 0
End: 100
Delta: 8

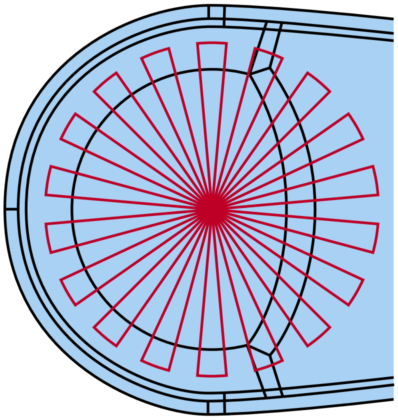
Pick center of spiral tool path

Tool path connections

Direction type
○ For&Back
● Forward

Connection type
● Stairs
○ Smooth

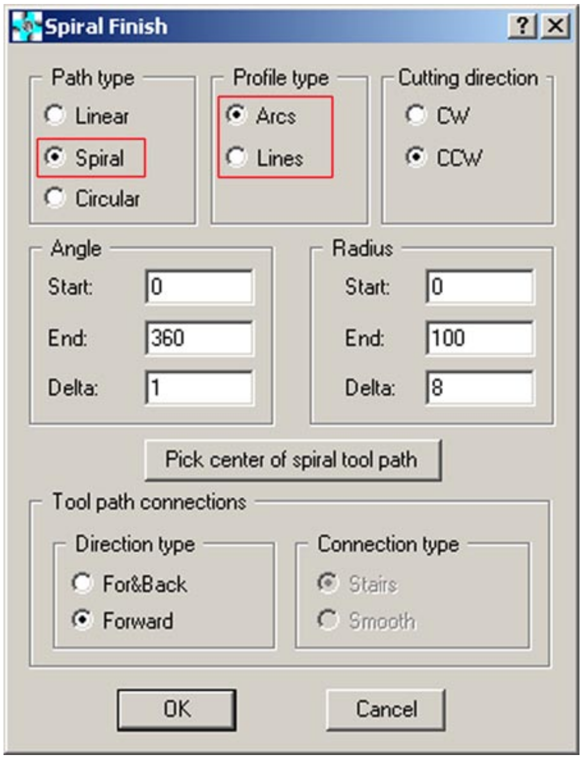
OK Cancel



Additional parameters:

- **Start Angle**
- **End Angle**
- **Delta Angle**

Path type: Spiral



Spiral Finish

Path type
○ Linear
● Spiral
○ Circular

Profile type
● Arcs
○ Lines

Cutting direction
○ CW
● CCW

Angle
Start: 0
End: 360
Delta: 1

Radius
Start: 0
End: 100
Delta: 8

Pick center of spiral tool path

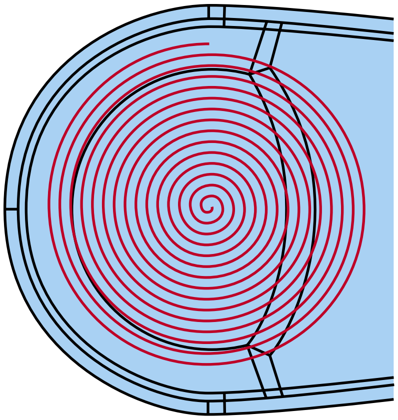
Tool path connections

Direction type
○ For&Back
● Forward

Connection type
● Stairs
○ Smooth

OK Cancel

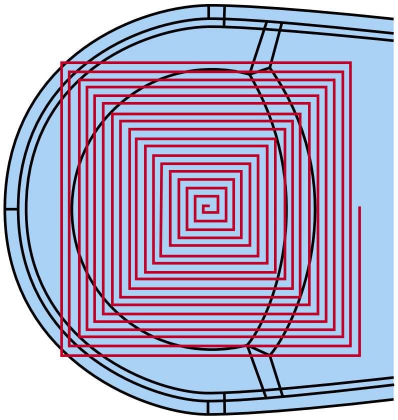
- **Profile type: Arcs**



Additional parameters:

- **Start Angle**
- **End Angle**
- **Start Radius**
- **End Radius**
- **Delta Radius**

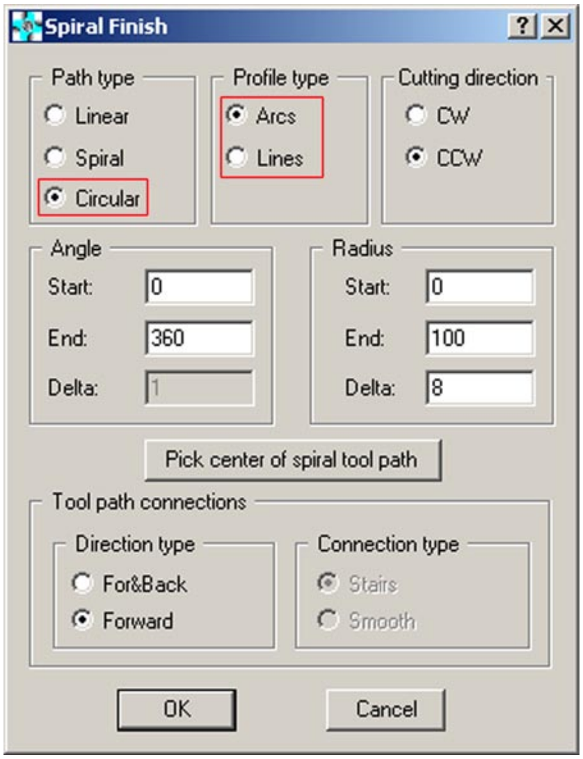
- **Profile type: Lines**



Additional parameters:

- **Start Radius**
- **End Radius**
- **Delta Radius**

Path type: Circular



Spiral Finish

Path type: ○ Linear ○ Spiral ● Circular

Profile type: ● Arcs ○ Lines

Cutting direction: ○ CW ● CCW

Angle
Start: 0
End: 360
Delta: 1

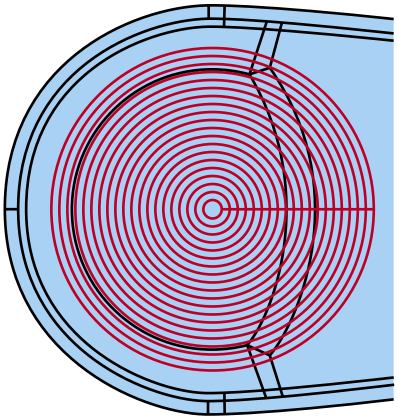
Radius
Start: 0
End: 100
Delta: 8

Tool path connections

Direction type: ○ For&Back ● Forward

Connection type: ● Stairs ○ Smooth

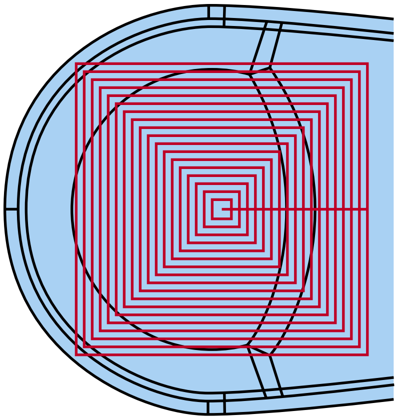
- **Profile type: Arcs**



Additional parameters:

  - **Start Angle**
  - **End Angle**
  - **Start Radius**
  - **End Radius**
  - **Delta Radius**

- **Profile type: Lines**



Additional parameters:

  - **Start Radius**
  - **End Radius**
  - **Delta Radius**

Cutting direction

The spiral will be machined in either clockwise (**CW**) or counterclockwise (**CCW**) direction. The direction also affects the angle area that will be machined. Machining always starts at the angle given in **Start angle** up to the **End angle** in the cutting direction.

Angle

You can limit the angle area that will be machined by setting values for the **Start** and **End** angle. The direction of the positive X-axis is entered with a value of 0 degrees. Machining will always start from the given start angle and will move in the cutting direction towards the end angle.

- The **Delta angle** is used to specify the angle steps of the tool path spiral in the **Path type: Linear** option.

Radius

You can set values for the **Start** and **End radius** of the tool path spiral.

- The **Delta radius** specifies the offset distance to the next circle or spiral circumference of the tool path spiral.

Pick center of tool path spiral

This button enables you to define the center of the tool path spiral that will be projected on the model. If you do not choose any center, the system uses the current Home position as the center of the spiral.

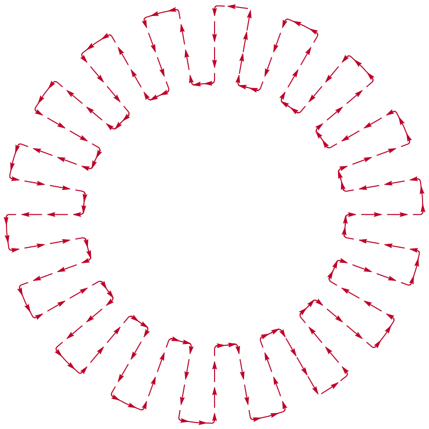
Enter the X,Y,Z coordinates of the center point into the Edit bar and confirm by clicking on Enter or define a point by pick.

Tool path connections

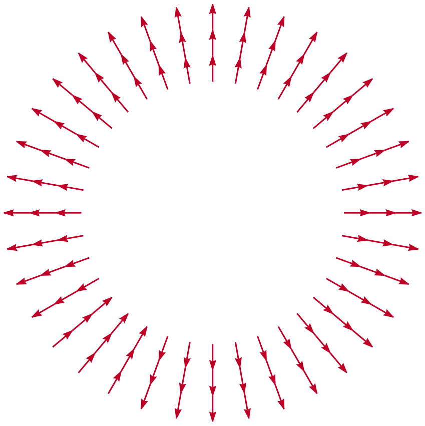
Direction type

- **For&Back**

  The tool finishes one line of the spiral pattern and then moves directly to the next tool path line and so on. It mills forward and backward without leaving the material - thus constantly changing between climb milling and conventional milling.



- **Forward**



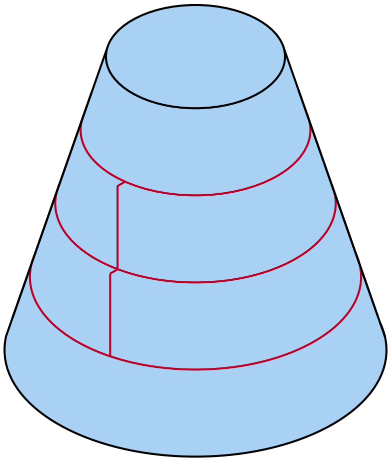
The tool moves finishes one line of the linear pattern. At the end of the line, the tool rapids (G0) to the safety distance and then to the start of the next cut line. The tool will always use climb or conventional milling.

Connection type

This option enables you to control the way the tool moves between two adjacent tool paths.

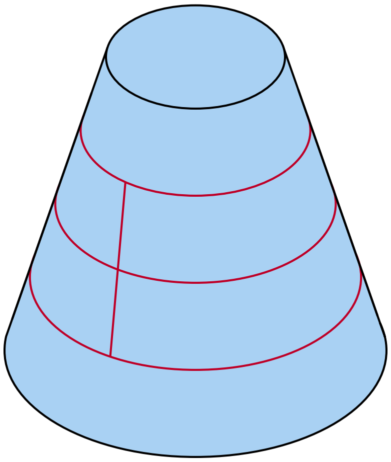
- **Stairs**

When the **Stairs** option is activated, the tool moves in two steps between two adjacent tool paths. If the next path is lower than the present path, the tool first moves away from the surface in the XY direction and then moves downwards in Z. In case the next path is higher than the present path, the tool will first move upwards in Z and it will then approach the surface in the XY direction.



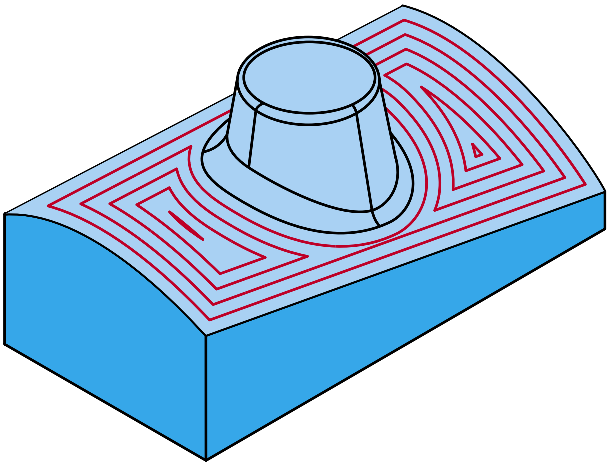
- **Smooth**

Between two adjacent tool paths, the tool moves on the surface of the model. The line connecting the two adjacent tool paths is projected on the model. When you use the **Smooth** option, the tool will not leave the material when moving to the start point of the next tool path.



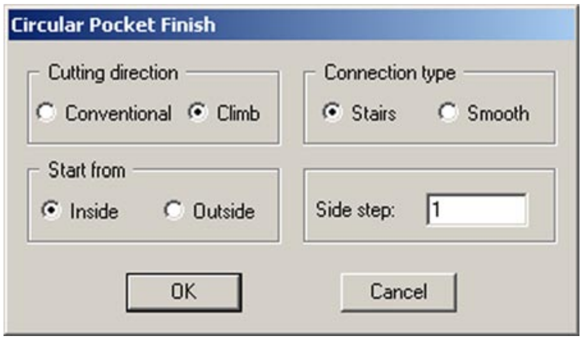
7.5.7 Circular Pocket Finish Strategy

This strategy uses the selected **Working area** to create a 2D pocket line pattern on a plane above the model. The line pattern is then projected on the 3D model. During this projection, the Z-position of the tool is calculated to avoid gouging of the material.



The shape of the **Working area** and the values for the „Tool position“ in the Working area dialog of this job will directly affect the tool path.

The **Circular Pocket Finish** dialog displays the parameters of the Circular Pocket Finish strategy..



Circular Pocket Finish

Cutting direction: ○ Conventional ● Climb

Connection type: ● Stairs ○ Smooth

Start from: ● Inside ○ Outside

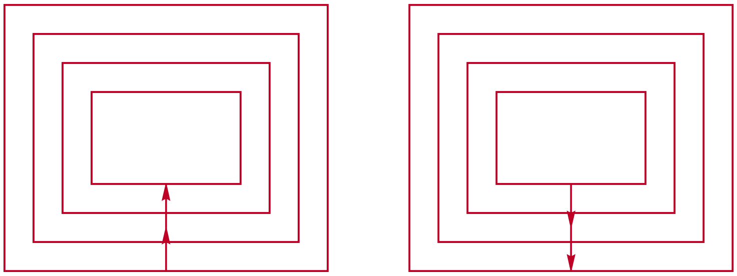
Side step: 1

Cutting direction

The pocket will be machined in either clockwise (**CW**) or counterclockwise (**CCW**) direction.

Start from

The machining can be started from **Outside** or **Inside** the pocket.



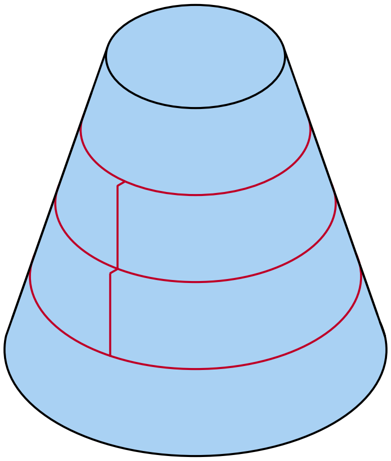
The combination of both fields, **Cutting direction** and **Start from**, will result in climb or conventional milling. Select **CCW/Outside** or **CW/Inside** to perform climb milling - any other combination will result in conventional milling of the model.

Connection type

This option enables you to control the way the tool moves between two adjacent tool paths.

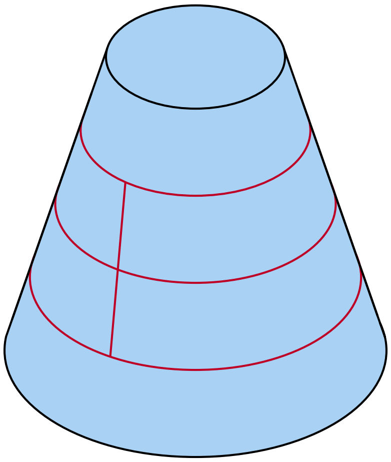
- **Stairs**

When the **Stairs** option is activated, the tool will move in two steps between two adjacent tool paths. If the next path is lower than the present path, the tool first moves away from the surface in the XY direction and then moves downwards in Z. In case the next path is higher than the present path, the tool will first move upwards in Z and it will then approach the surface in the XY direction.



- **Smooth**

Between two adjacent tool paths, the tool moves on the surface of the model. The line connecting the two adjacent tool paths is projected on the model. When you use the **Smooth** option, the tool will not leave the material when moving to the start point of the next tool path.

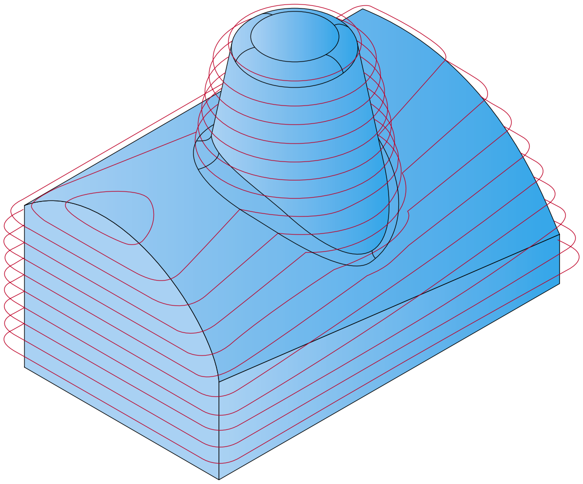


Side Step

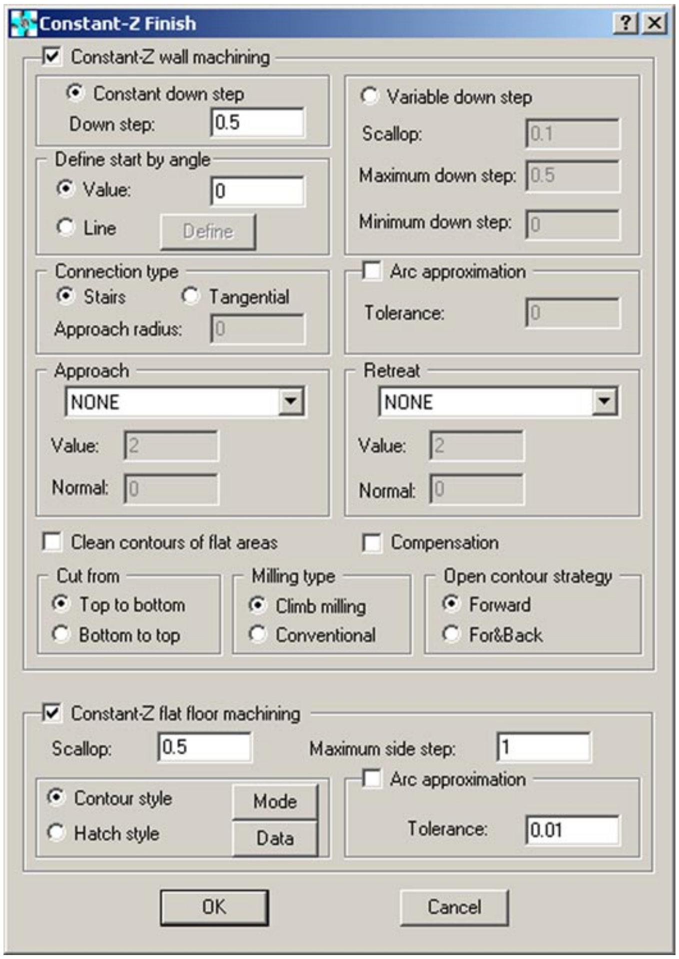
In pocket finishing, SolidCAM generates the pocket line pattern on a 2D plane above the model and then projects it on the 3D model. The side step value determines the constant distance between the lines of the pocket created on the 2D plane before it will be projected on the model.

7.5.8 Constant-Z Finish Strategy

This strategy produces very high quality finish especially on steep surfaces and minimizes full 3 axis movements that slow down older machines and controllers. The scallop option adjusts the necessary Z-down steps to the surface flow and results in shorter machining times.



The **Constant-Z Finish** dialog displays the parameters of the **Constant Z Finish**.



Constant-Z Finish

☑ Constant-Z wall machining

● Constant down step
Down step: 0.5

○ Variable down step
Scallop: 0.1
Maximum down step: 0.5
Minimum down step: 0

Define start by angle
● Value: 0
○ Line

Connection type
● Stairs ○ Tangential
Approach radius: 0

☐ Arc approximation
Tolerance: 0

Approach
NONE
Value: 2
Normal: 0

Retreat
NONE
Value: 2
Normal: 0

☐ Clean contours of flat areas
☐ Compensation

Cut from
● Top to bottom
○ Bottom to top

Milling type
● Climb milling
○ Conventional

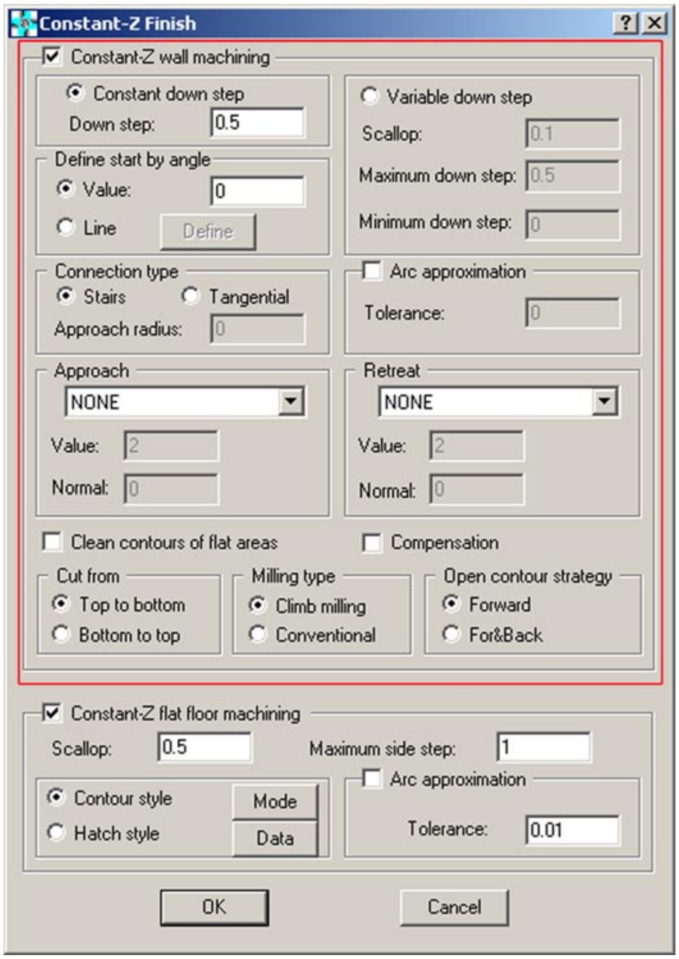
Open contour strategy
● Forward
○ For&Back

☑ Constant-Z flat floor machining
Scallop: 0.5
Maximum side step: 1

● Contour style
○ Hatch style

☐ Arc approximation
Tolerance: 0.01

Constant-Z wall machining



Constant-Z Finish

☑ Constant-Z wall machining

● Constant down step
Down step: 0.5

○ Variable down step
Scallop: 0.1
Maximum down step: 0.5
Minimum down step: 0

Define start by angle
● Value: 0
○ Line

Connection type
● Stairs ○ Tangential
Approach radius: 0

☐ Arc approximation
Tolerance: 0

Approach
NONE
Value: 2
Normal: 0

Retreat
NONE
Value: 2
Normal: 0

☐ Clean contours of flat areas
☐ Compensation

Cut from
● Top to bottom
○ Bottom to top

Milling type
● Climb milling
○ Conventional

Open contour strategy
● Forward
○ For&Back

☑ Constant-Z flat floor machining
Scallop: 0.5
Maximum side step: 1

● Contour style
○ Hatch style

☐ Arc approximation
Tolerance: 0.01

Activate the checkbox to machine all areas of the model that are not parallel to the XY-plane, i.e. are not planar. The Z-down step can be controlled by two different options:

Down step options

SolidCAM enables you to choose one of two options for Constant-Z wall machining:

- **Constant down step**

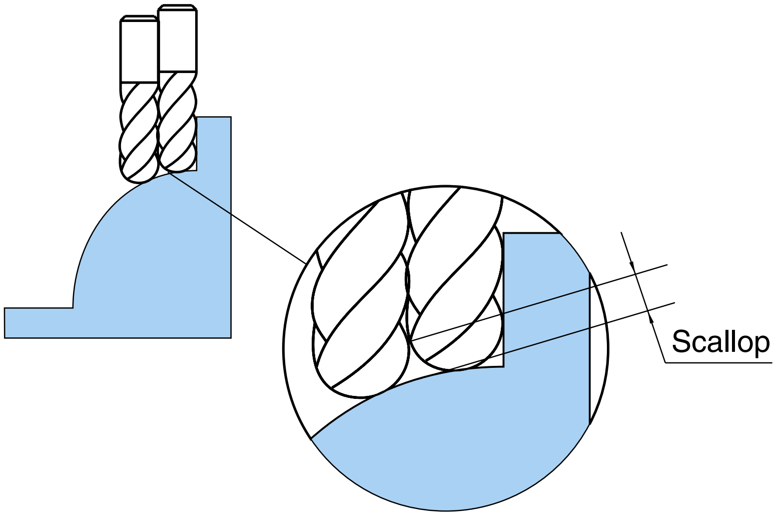
  Starting from the **Job Upper level**, SolidCAM will use the specified down step to calculate each successive Z-level. A value of 20 for the Job Upper plane and a constant down step of 1.5 will produce a tool path on the model contour at Z 20, Z18.5, Z17, Z 15.5, ... etc. until the Job Lower plane has been reached.

- **Variable down step**

  SolidCAM will automatically calculate the necessary down steps to produce the specified scallop (cusp height) on the model surface. On steep areas, the down step will naturally be larger than on more level areas.

- **Scallop**

The **Scallop** value determines the cusp height of the resulting surface finish. The smaller the value, the smoother the surface finish will be.



Maximum down step

On vertical walls of the model, SolidCAM could apply large down steps as this would theoretically result in **0** scallop. To avoid large cutting depth and tool load, you can limit the maximum step between two successive Z-levels.

Minimum down step

On nearly level areas, the necessary down step to achieve the specified scallop could be very small. You can set a minimum down step value that will override the smaller down step generated by the scallop calculation.

Start Machining (Define start by angle)

- **Angle**

Enter the angle that defines the start position of the tool path. A value of 0 will generate the tool path start point on the intersection of the contour and the positive X-axis.

- **Line**

When you choose the **Line** option, the **Define** button will be activated and you can define a line that determines the orientation of the tool path. The direction vector is defined by picking two points that define the start point and end point of the direction line. The angle of the selected line will be output to the angle field.

Arc approximation

This command enables you to create G2/G3 G-Code output from **Constant Z** operations. SolidCAM checks whether successive points of the calculated tool path can be connected using an arc or circle. If an arc or circle connection within the specified arc approximation tolerance can be made, you will receive arc and circle interpolation commands G2 and G3 in the generated G-Code.

This feature can drastically reduce the number of lines in G-Code files. Most CNC-controllers and machines work much faster on arcs and circles than on single tool path points or splines. Arc approximation will increase actual feed rates on older CNC-machines and the machine will work smoother.

The **Tolerance** value defines the tolerance SolidCAM uses to position tool path points on arcs or circles.

The arc approximation value should be smaller than the specified value for the surface offset. A warning message will be displayed if a larger value could cause gouging of the model.

Approach/Retreat

You can control the way the tool approaches and retreats from the profile. Select an option from the list field and enter the **Distance value** for this option.

SolidCAM enables you the following optionsfor the **Approach/Retreat**:

- **None**
- **Normal**
- **Arc**
- **Tangent**

For more information on the Approach/Retreat option, please refer to the topic **6.1.2**.

Compensation

This option enables SolidCAM to use the tool radius compensation options G4x of the CNC controller in G-Code output.

Clean contours of flat areas

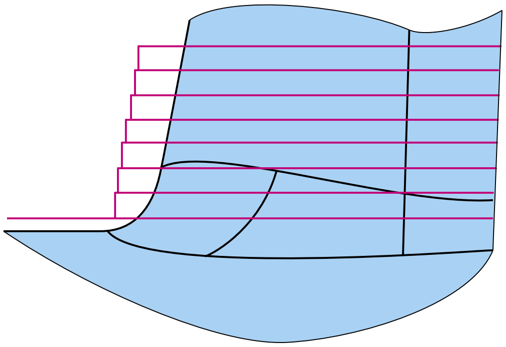
In prismatic solid parts, if there is a ledge at a depth that falls between down-steps or a wall at the edge of a flat bottom of Constant-Z machining, SolidCAM can still recognize this ledge or wall and machine it as if in a profile job (between the machining of the previous and next Z-steps).

Connection type

This option enables you to control the way the tool moves between two adjacent tool paths.

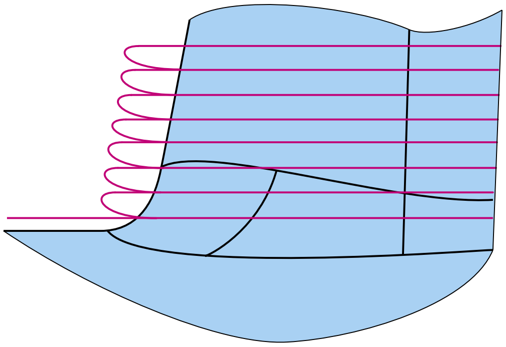
- **Stairs**

With the **Stairs** tool path connection, the tool will move in two steps between two adjacent tool paths. If the next path is lower than the present path, the tool first moves away from the surface in the XY direction and then moves downwards in Z. In case the next path is higher than the present path, the tool will first move upwards in Z and then it will approach the surface in the XY direction.



- **Tangential**

Between two adjacent tool paths, the tool moves away from the contour in a tangent movement. It continues in a 3D arc/spiral movement to the next Z-level and then approaches the contour in another tangent movement.



The **Tangential** tool path connection smooths lead in/out moves and is especially useful for High Speed Machining. It will avoid „Witness marks“ on the model that could be caused from a Z approach movement on the model‘s surface.

The **Approach radius** defines the radius of the 3D arc that is used to create the tangential move between two successive Z-levels. The value is the maximum radius used as SolidCAM will automatically reduce the radius if the model topology will not leave enough space to approach with the specified radius. This could happen in narrow cavities of the model when a large tool is being used.

Cutting direction (Cut from:)

- **Top to bottom**

  The machining will start on the Job Upper plane and continue downward in the -Z direction until the Job Lower plane is reached. The tool finishes one line of the linear pattern, moves directly to the next line and so on. It mills forward and backward without leaving the material - thus constantly changing between climb milling and conventional milling.

- **Bottom to top**

  The tool will start from the Job Lower plane and machine towards the Job Upper plane.

Milling type

- **Climb milling**

  The tool cuts on the left side of the model contour. Unless the selected **Open contour strategy** is set to **For&Back**, contours will always machine in the climb milling direction.

- **Conventional**

  The tool will cut on the right side of the model contour. On outside contours, this will generate a tool path in clockwise direction.

Open contour strategy

This field controls the milling direction on contours that cannot be closed because the model or a specified working area does not enable SolidCAM to continue the tool path in a closed loop.

- **Forward**

  The tool will retreat to the clearance plane and move to the start point of the next tool path, always keeping the cutting direction specified in the **Milling type**.

- **For&Back**

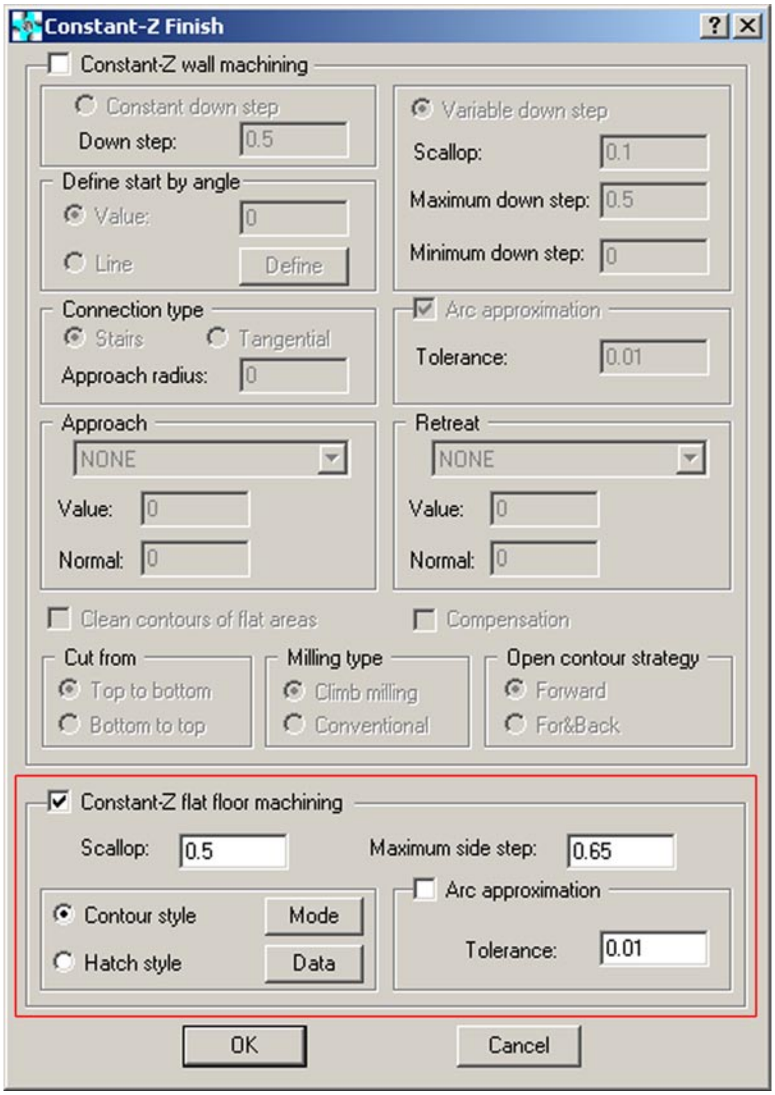
  This option enables the tool to change the cutting direction when it encounters open contours on the model. The milling direction on open contours will change between climb and conventional milling until the open contour has been finished. With the **For&Back** option, rapid moves from Z-Level to Z-level will be greatly reduced.

Fillet size for last cut

This option enables you to add a radius to sharp corners of the tool path without having to change the geometry.

For more information on this option, please refer to the topic **6.1.4**.

Constant-Z flat floor machining



Constant-Z Finish

☐ Constant-Z wall machining

○ Constant down step

Down step: 0.5

Define start by angle

● Value: 0

○ Line

● Variable down step

Scallop: 0.1

Maximum down step: 0.5

Minimum down step: 0

Connection type

● Stairs ○ Tangential

Approach radius: 0

☑ Arc approximation

Tolerance: 0.01

Approach

NONE

Value: 0

Normal: 0

Retreat

NONE

Value: 0

Normal: 0

☐ Clean contours of flat areas

☐ Compensation

Cut from

● Top to bottom

○ Bottom to top

Milling type

● Climb milling

○ Conventional

Open contour strategy

● Forward

○ For&Back

☑ Constant-Z flat floor machining

Scallop: 0.5

Maximum side step: 0.65

● Contour style

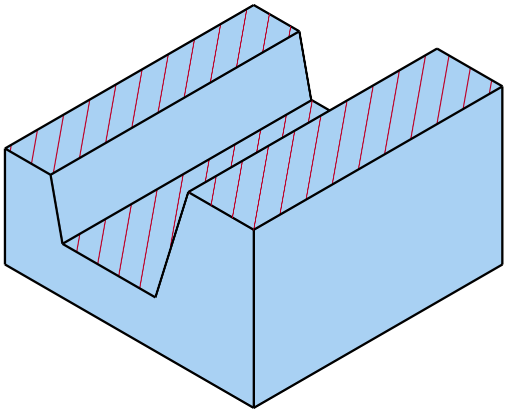
○ Hatch style

☐ Arc approximation

Tolerance: 0.01

Use the checkbox to activate this option. SolidCAM recognizes planar areas on 3D models. For flat, planar areas of the model, working areas will be automatically created that can be machined using a hatch or contour style pocket.

Flat floor machining greatly reduces programming time of 3D models with 2.5D features like pockets and profiles on different Z-levels.



Scallop

Enter the scallop (cusp height) of the finished surface you want to achieve. SolidCAM will automatically calculate the side step of the tool.

- **Maximum side step**

  If you use a flat tool or a bull nose cutter, the side step will be as large as the flat area of the tool, e.g. for a tool diameter 6 with a corner radius of 1, the side step on flat areas would be 4. To limit the cutting width of the tool, enter the maximum side step value.

Machining style

SolidCAM enables you to choose two styles of the floor machining during Constant-Z finishing:

Contour style

Select this option if you want to machine the flat floor areas on your model as a contour style pocket. The pocket tool path will travel at parallel offsets to the outside contour of the flat area.

1. Choose the **Contour** strategy with the radio button.
2. Click on the **Data** button to define the Contour parameters.

For more information on the **Contour machining**, please refer to the topic **6.2.5**.

Hatch style

The tool mills the flat area in a linear pattern.

1. Choose the **Hatch** strategy with the radio button.
2. Click on the **Data** button to define the Hatch parameters.

For more information on the **Hatch machining**, please refer to the topic **6.2.4**.

Withthe **Mode** button SolidCAM enables you to locate open pocket areas and enables you to machine them with a specific strategy. This strategy enables you to combine pocket and profile tool paths in order to provide the best productivity in open pocket machining..

For more information on the **Open Pocket machining**, please refer to the topic **6.2.5**.

Arc approximation

This command enables you to create G2/G3 G-Code output from **Constant Z** operations. SolidCAM checks whether successive points of the calculated tool path can be connected using an arc or circle. If an arc or circle connection within the specified arc approximation tolerance can be made, you will receive arc and circle interpolation commands G2 and G3 in the generated G-Code.

This feature can drastically reduce the number of lines in G-Code files. Most CNC-controllers and machines work much faster on arcs and circles than on single tool path points or splines. Arc approximation will increase actual feed rates on older CNC-machines and the machine will work smoother.

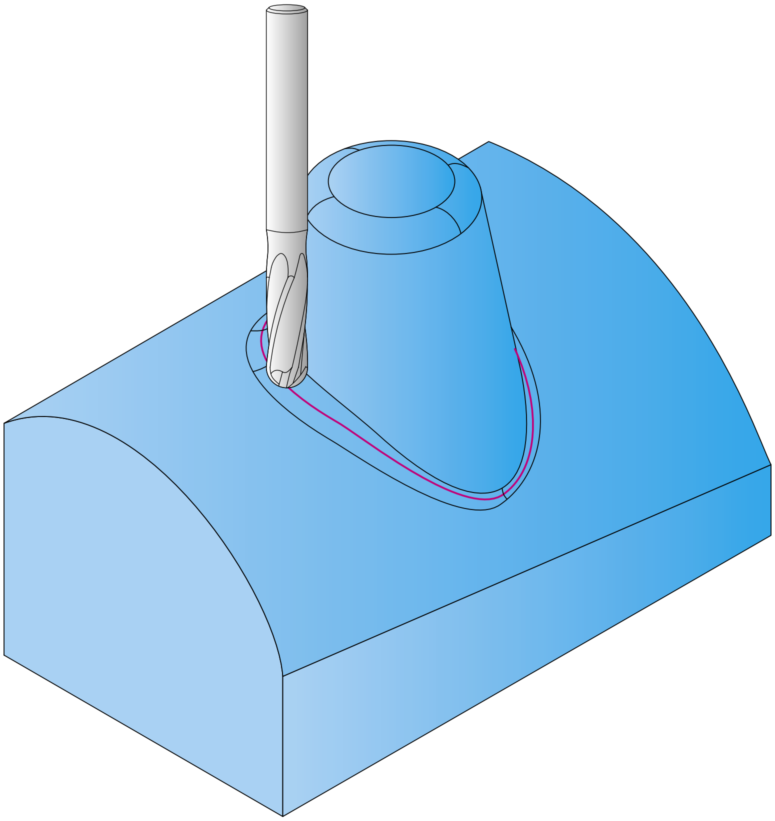
The **Tolerance** value defines the tolerance SolidCAM uses to position tool path points on arcs or circles.

Fillet size for last cut

This option enables you to add a radius to sharp corners of the tool path without having to change the geometry.

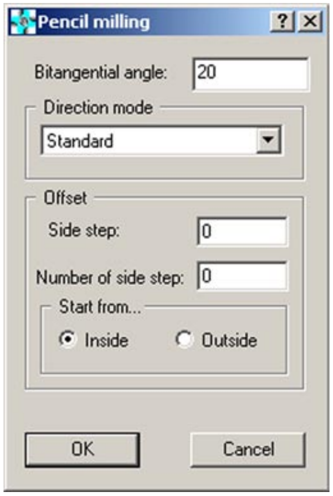
For more information on this option, please refer to the topic **6.1.4**.

7.5.9 Pencil milling



Pencil milling moves the cutter along internal corners, removing machining cusps left by previous machining operations. By allowing the tool to move in all 3 axes simultaneously, it is possible to pencil mill complex 3D profiles.

The **Pencil milling dialog** enables you to define the parameters of the Pencil milling.



Pencil milling

Bitangential angle: 20

Direction mode

Standard

Offset

Side step: 0

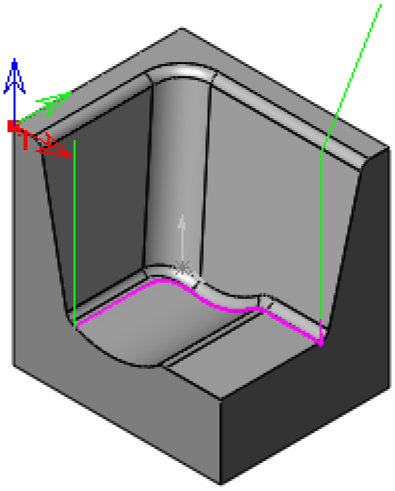
Number of side step: 0

Start from...

● Inside ○ Outside

Threshold angle

The threshold angle is the minimum angle value of the internal corner. SolidCAM will machine corners from the threshold angle parameter or any angle greater than this value. For each corner SolidCAM calculates the center line to machine the angle area optimally.



Offset

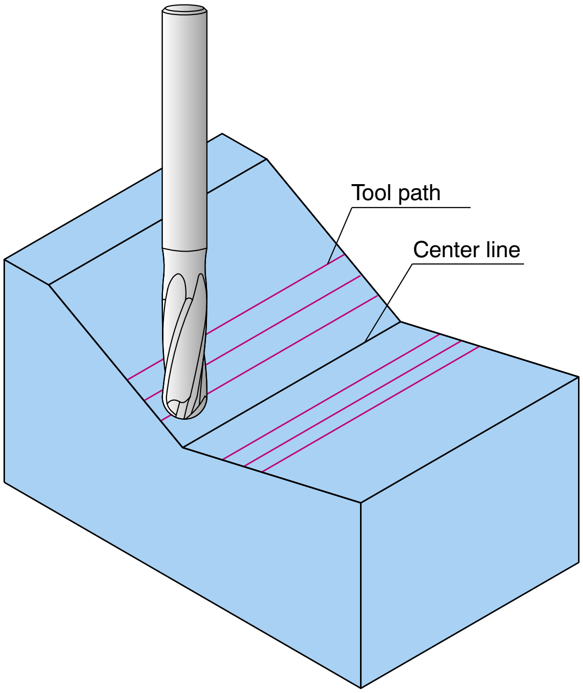
SolidCAM enables you to machine the area defined by the threshold angle with the constant offset from both sides of the center line.

- **Side step**

  Defines the side step of both sides of the original center line of the **Pencil milling**.

- **Number of side steps**

  This option enables you to determine the number of side steps.



Start from:

- **Inside**

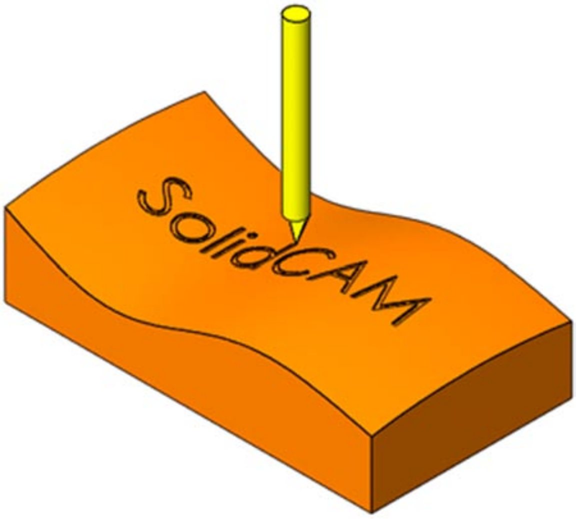
  This allows you to start the side steps from the tool path outwards.

- **Outside**

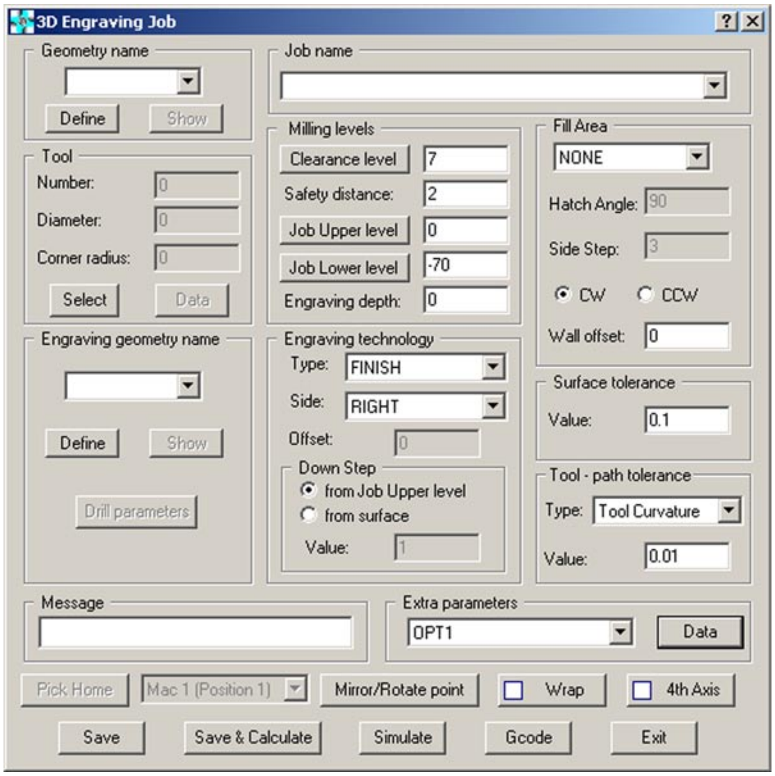
  This allows you to start from outside side step working toward the original tool path.

7.6 3D Engraving Job

You can mill text or any other profile on a 3D geometry. The profile will be projected on the surface, engraving the contour at a specified depth.



When you select the commands **Add>3D Engraving**, the **3D Engraving Job** screen will be displayed.


3D Engraving Job
Geometry name
Define
Show
Job name
Milling levels
Fill Area
Tool
Select
Data
Engraving geometry name
Define
Show
Drill parameters
Engraving technology
Down Step
Surface tolerance
Tool - path tolerance
Message
Extra parameters
Data
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

Tool
Number: 0
Diameter: 0
Corner radius: 0

Milling levels
Clearance level 7
Safety distance: 2
Job Upper level 0
Job Lower level -70
Engraving depth: 0

Fill Area: NONE
Hatch Angle: 90
Side Step: 3
● CW ○ CCW
Wall offset: 0

Engraving technology
Type: FINISH
Side: RIGHT
Offset: 0
Down Step
● from Job Upper level
○ from surface
Value: 1

Surface tolerance
Value: 0.1

Tool - path tolerance
Type: Tool Curvature
Value: 0.01

Extra parameters: OPT1

Mac 1 (Position 1)
☐ Wrap
☐ 4th Axis

7.6.1 Geometry name

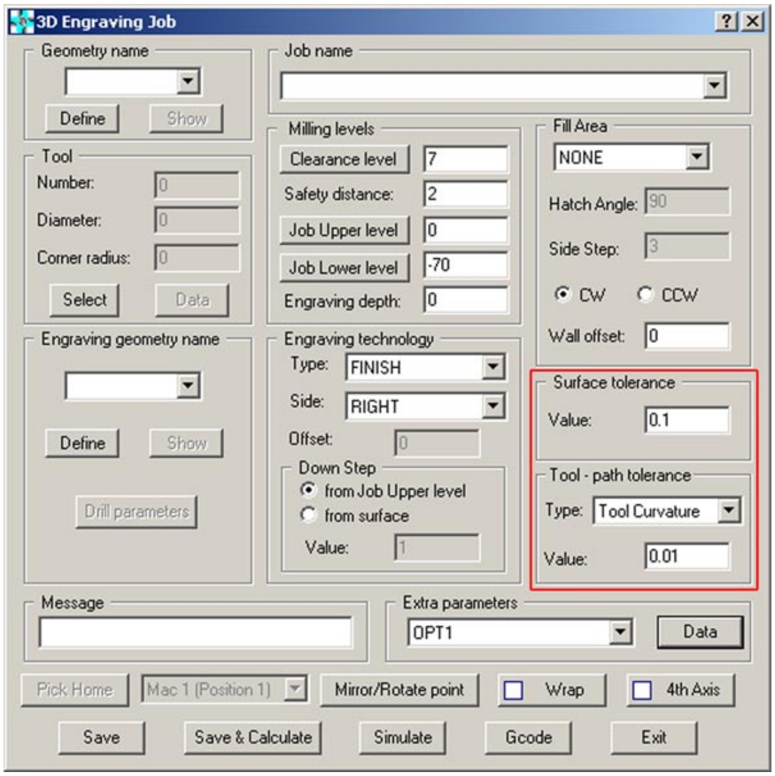
In this section specify the 3D model, i.e. the solids and surfaces on which you want to engrave profiles or text.

For more information on **3D Model Geometry** , please refer to the topic **4.4**

7.6.2 Engraving depth

This parameter specifies the depth the tool will penetrate the surface of the 3D model.

7.6.3 Tolerance


3D Engraving Job
Geometry name
Define
Show
Job name
Tool
Select
Data
Milling levels
Fill Area
Engraving geometry name
Define
Show
Drill parameters
Engraving technology
Down Step
Message
Extra parameters
Data
Pick Home
Mirror/Rotate point
☐ Wrap
☐ 4th Axis
Save
Save & Calculate
Simulate
Gcode
Exit

Tool
Number: 0
Diameter: 0
Corner radius: 0

Milling levels
Clearance level: 7
Safety distance: 2
Job Upper level: 0
Job Lower level: -70
Engraving depth: 0

Fill Area: NONE
Hatch Angle: 90
Side Step: 3
● CW ○ CCW
Wall offset: 0

Engraving technology
Type: FINISH
Side: RIGHT
Offset: 0
Down Step
● from Job Upper level
○ from surface
Value: 1

Surface tolerance
Value: 0.1

Tool - path tolerance
Type: Tool Curvature
Value: 0.01

Extra parameters: OPT1

Mac 1 (Position 1)

The tolerance settings will affect the surface quality of the engraving operation.

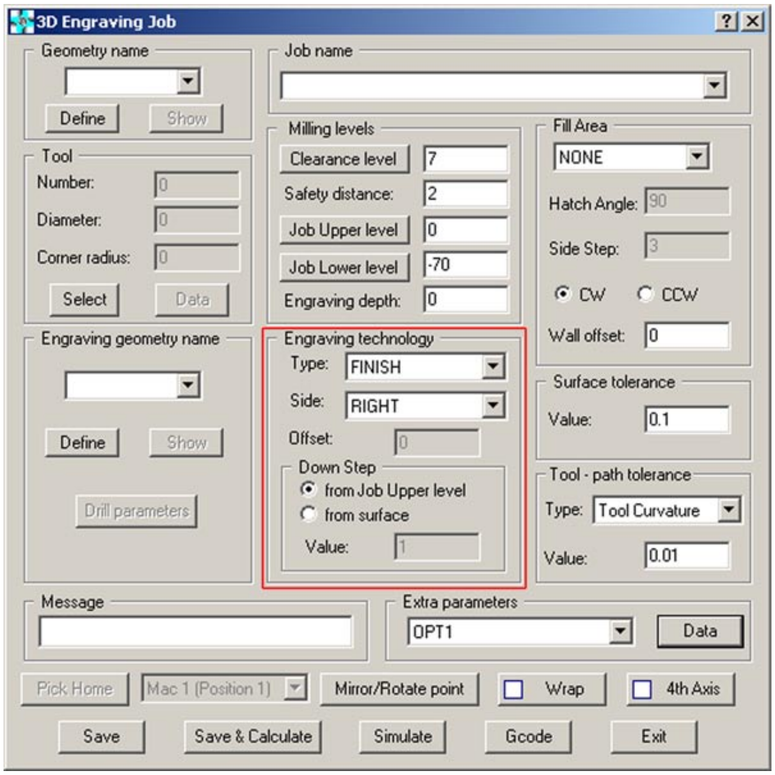
For more information on **Tolerances**, please refer to the topic **7.3**

7.6.4 Engraving geometry name

In this section you have to specify the contour, i.e. profile geometry that you want to engrave on the 3D model surface.

For more information on **Profile Geometry**, please refer to the topic **4.5**

7.6.5 Engraving technology


3D Engraving Job
Geometry name
Define
Show
Job name
Select
Data
Engraving geometry name
Define
Show
Drill parameters
Message
Extra parameters
Data
Pick Home
Mirror/Rotate point
Wrap
4th Axis
Save
Save & Calculate
Simulate
Gcode
Exit

Tool
Number: 0
Diameter: 0
Corner radius: 0

Milling levels
Clearance level: 7
Safety distance: 2
Job Upper level: 0
Job Lower level: -70
Engraving depth: 0

Fill Area: NONE
Hatch Angle: 90
Side Step: 3
● CW ○ CCW
Wall offset: 0

Engraving technology
Type: FINISH
Side: RIGHT
Offset: 0
Down Step
● from Job Upper level
○ from surface
Value: 1

Surface tolerance
Value: 0.1

Tool - path tolerance
Type: Tool Curvature
Value: 0.01

Extra parameters: OPT1

Mac 1 (Position 1)

☐ Wrap ☐ 4th Axis

Type

This option enables you to semi-finish and/or finish the engraved profile on the translated surface.

- **Semi-finish** will prepare the engraving profile to an offset specified in the Offset field with cutting depths given in the Down step field.
- **Finish** will machine the engraving profile to its final dimensions in one down step.
- **Both**. Use this option to machine the limit first with a semi-finish cut and then with a finishing cut.

Side

The tool can cut on the left to the profile, right to the profile or on the engraving geometry.

- **Right**. The tool will cut on the right side of the profile geometry up to a distance defined by the Offset value.
- **Left**. The tool will cut on the left side of the profile geometry at a distance defined by the Offset value.
- **Middle**. The center of the tool will move on the profile geometry.

Offset

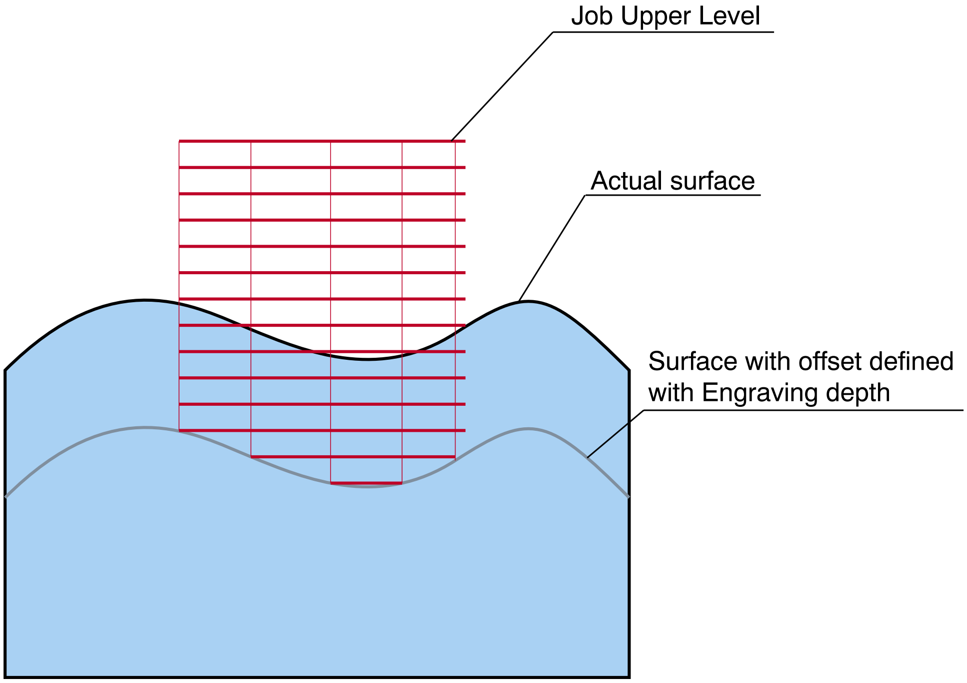
You can specify the offset from the engraving geometry for a Semi finish operation on the left or right side of the geometry. This offset will be removed by the Finish cut, if you had defined one.

Down step

This value defines the successive Z-cutting depth during the Semi-finish operation of the engraving profile.

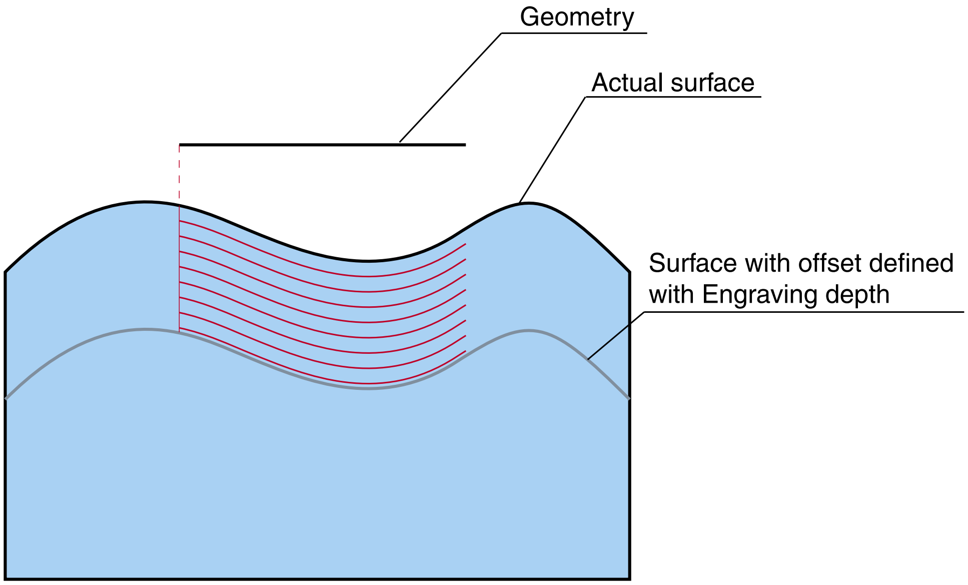
- **From Job Upper level**

  With this option SolidCAM does Roughing passes, at each down step, from the Job Upper Level to the virtual surface located at an offset from the actual surface. This offset is defined with the Engraving depth.



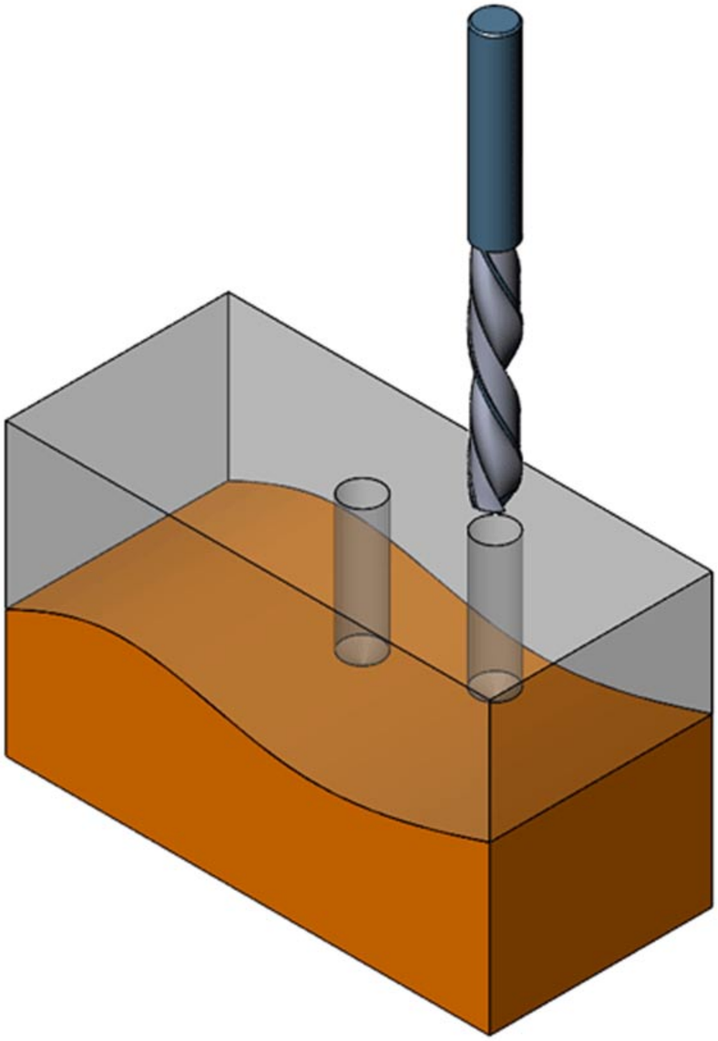
- **From surface**

In this case SolidCAM executes roughing passes, at each down-step, only from an actual surface to a virtual surface defined with engraving depth.

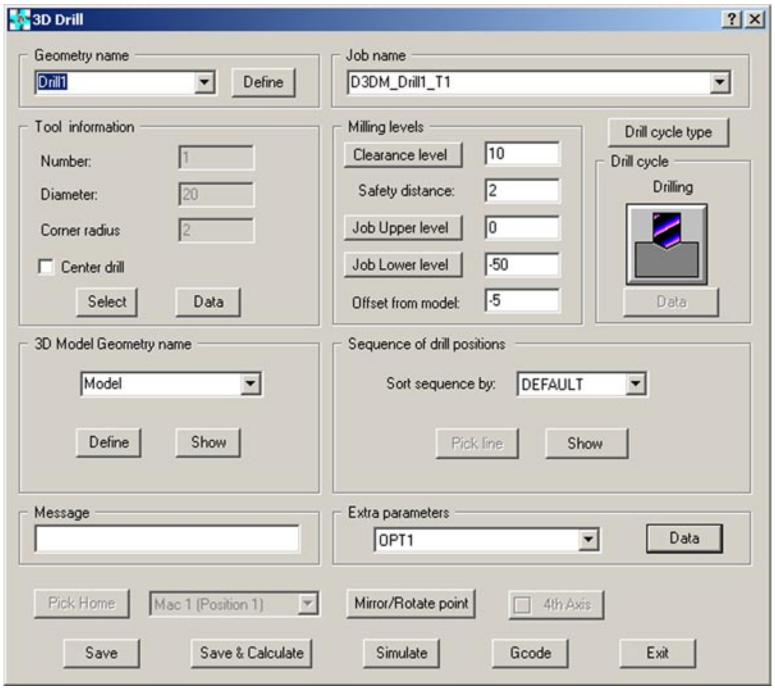

Geometry
Actual surface
Surface with offset defined with Engraving depth

7.7 3D Drill Job

This job enables you to perform drills and other canned drill cycles. In contrast to **Drill Job** (see the topic **6.3**), **3D Drill Job** enables you to take into account the solid model geometry. SolidCAM enables you to use holes, machined by this job, for the descent during 3D model roughing (for more information on the Descent statefies, please refer to the topic **7.4.6**). Like Drill Job, 3D Drill Job supports the canned drill cycles provided by your particular CNC Machine such as threading, peck, ream, boring etc...



When you select the commands **Add>3D Drill**, the **3D Drill Job** screen will be displayed.


3D Drill
Define
Tool information
Select
Data
3D Model Geometry name
Define
Show
Message
Milling levels
Clearance level
Job Upper level
Job Lower level
Drill cycle type
Data
Sequence of drill positions
Pick line
Show
Extra parameters
Data
Pick Home
Mirror/Rotate point
Save
Save & Calculate
Simulate
Gcode
Exit

Geometry name: Drill1

Job name: D3DM_Drill1_T1

Number: 1

Diameter: 20

Corner radius: 2

☐ Center drill

Clearance level: 10

Safety distance: 2

Job Upper level: 0

Job Lower level: -50

Offset from model: -5

Drill cycle: Drilling

3D Model Geometry name: Model

Sort sequence by: DEFAULT

Extra parameters: OPT1

Mac 1 (Position 1)

☐ 4th Axis

7.7.1 Geometry name

In this section, specify the drill positions, i.e. the drill geometry you want to use.

7.7.2 Tool

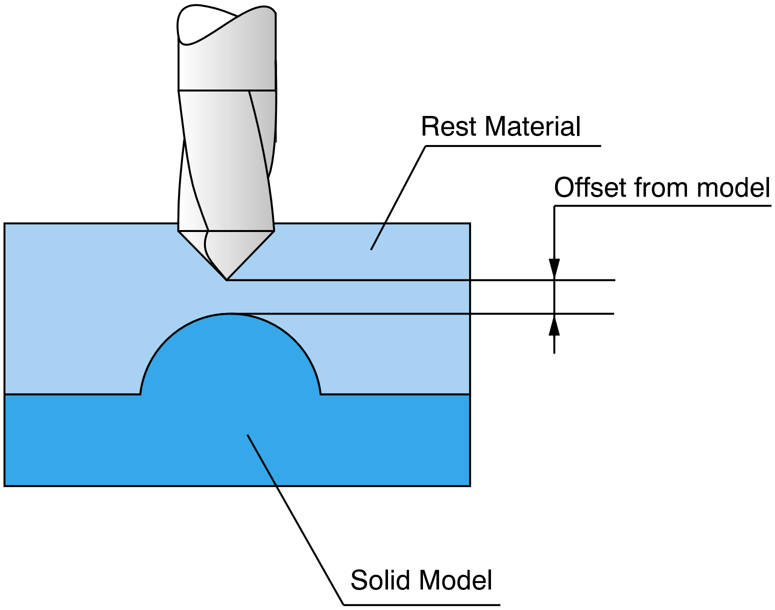
In addition to the standard tool options described in the Tool area section, SolidCAM enables you to label a tool as a center drill. This option is used for automatic sorting. In automatic sorting, tools that are labeled as center drills can be automatically moved to the top of the job list.

7.7.3 3D Model Geometry name

SolidCAM enables you to define the 3D model geometry. The 3D model geometry is used for the drilling depth definition.

7.7.4 Offset from model

SolidCAM enables you to define the depth of the 3D drilling by the Offset from model value. The Offset from model value can be either positive or negative. With the negative value SolidCAM enables the tool to go deeper into the solid model.



7.7.5 Drill cycle type

Click on the **Drill cycle** type button. The **Drill canned cycles** supported by the post-processor of the current CNC-controller will be displayed. Click on one of the icons to select the canned cycle you want to use in this job.

For more information on the **Drill cycles**, please refer to the topic **6.3.4**.

7.7.6 Sequence of drill positions

You can select one of 3 available options to control the sequence of the drill positions.

- **Default**

  The drills will be performed according to the order in which the drill points were picked during the geometry definition.

- **Line**

  You have to define a direction by selecting two points on your model.

- **Distance**

  This option minimizes the length of the necessary tool movement.

  For more information on the **Drill sequence**, please refer to the topic **6.3.5**.

Simulation

8

You can use the simulation to check and view the generated tool path after you have defined and calculated your machining operations.

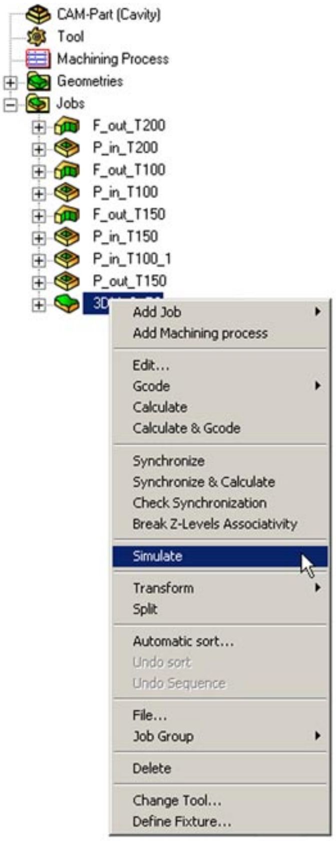
If you made mistakes in defining the jobs or used unsuitable milling strategies, the simulation can help you avoid problems you would otherwise experience during the actual production run.

- ❐ HostCAD Simulation mode
- ❐ 2D Simulation mode
- ❐ 3D Simulation mode
- ❐ VerifyPlus Simulation mode
- ❐ SolidVerify Simulation mode
- ❐ Rest Material Simulation mode

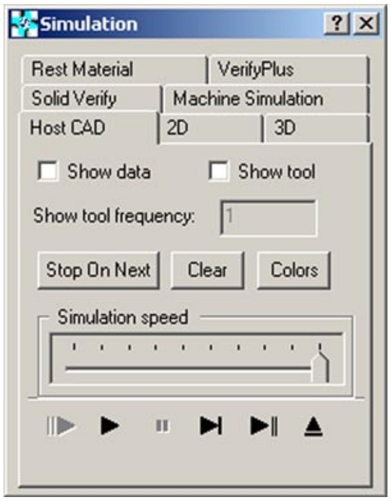
8.1 Introduction

The SolidCAM simulation option enables you to check and view the generated tool path after you have defined and calculated your machining operations. The simulation can help you avoid problems such as mistakes in defining the jobs or selection of an unsuitable milling strategy you would otherwise experience during the actual production run.

To simulate a Job, move the mouse **on the Job** and press the **right** mouse button in the CAM Manager. The **Job menu** will be displayed. Click on the **Simulate** option.


CAM-Part (Cavity)
Tool
Machining Process
Geometries
Jobs
F_out_T200
P_in_T200
F_out_T100
P_in_T100
F_out_T150
P_in_T150
P_in_T100_1
P_out_T150
3D
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...

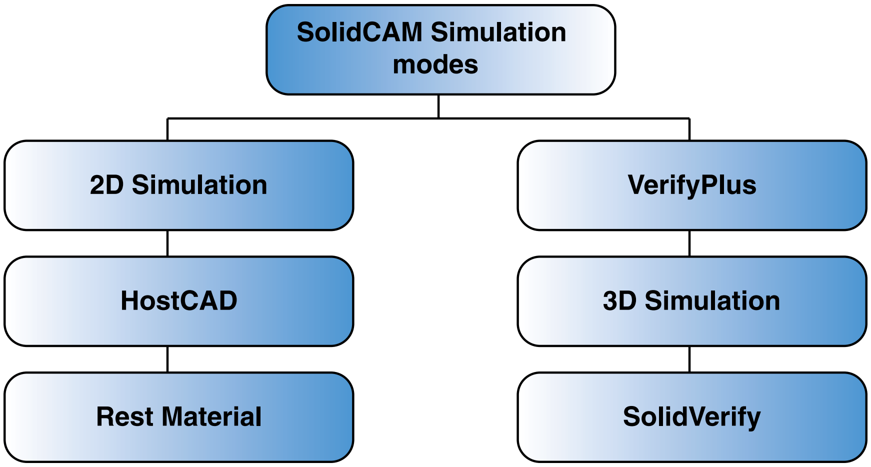
1. You also can simulate a group of selected Jobs. After you select the Jobs with the **Control -**key, right click on them and choose the **Simulation** option from the menu.
2. To simulate all the Jobs, right click on the **Jobs** field in the **SolidCAM Manager** and choose the **Simulation** option from the menu.


Simulation
Rest Material
VerifyPlus
Solid Verify
Machine Simulation
Host CAD
2D
3D
☐ Show data
☐ Show tool
Show tool frequency:
1
Stop On Next
Clear
Colors
Simulation speed

The **Simulation dialog** box will be displayed.

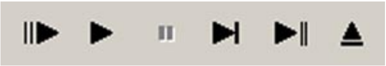
8.1.1 Simulation modes

SolidCAM offers various simulation modes:


SolidCAM Simulation modes
2D Simulation
HostCAD
Rest Material
VerifyPlus
3D Simulation
SolidVerify

8.1.2 Simulation Controls

The **Simulation controls** are used to play, pause and exit the simulation.



Click on the **control buttons** to run and stop the simulation:

||▶ **Turbo mode**. SolidCAM makes the simulation in the computer's memory without showing it on the screen. The image will be shown when the simulation is completed or when the **Pause** button is pressed.

▶ **Plays** the simulation in continuous mode.

II **Pauses** the simulation when in continuous play mode.

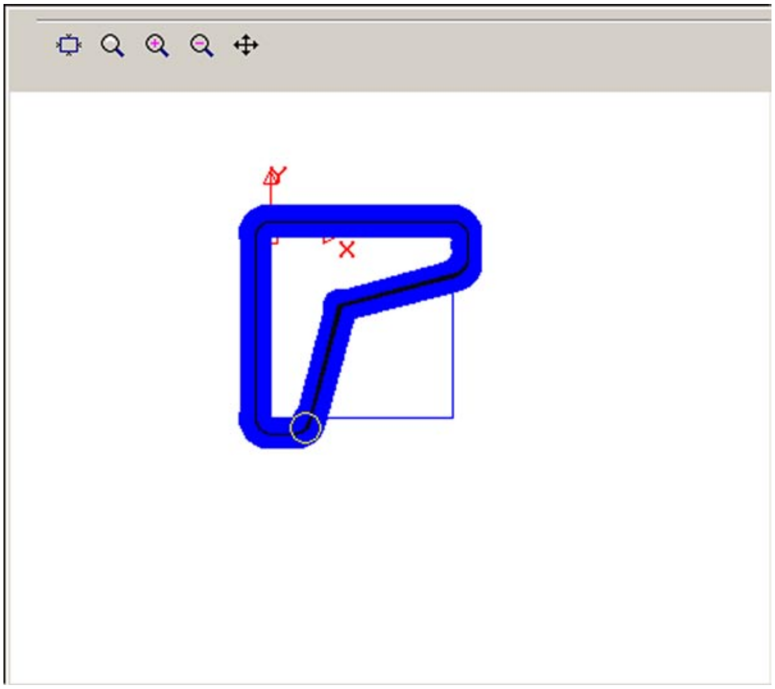
▶| **Single Step** mode (simulate the next tool movement by clicking on the symbol or by using the space bar of your keyboard).

▶|| **Job step mode.** Simulation is shown separately Job by Job.

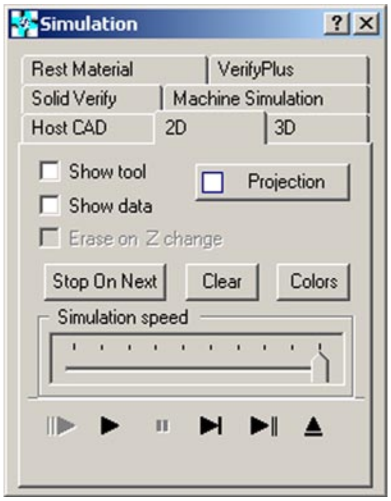
⏏ **Exits** from the simulation module.

8.2 2D simulation mode

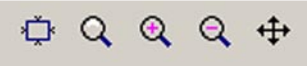
This simulation mode enables you to display the tool path in the projection view. This feature is useful in prismatic parts machining.



8.2.1 Simulation Control


Simulation
Rest Material
VerifyPlus
Solid Verify
Machine Simulation
Host CAD
2D
3D
☐ Show tool
☐ Show data
☐ Erase on Z change
Projection
Stop On Next
Clear
Colors
Simulation speed

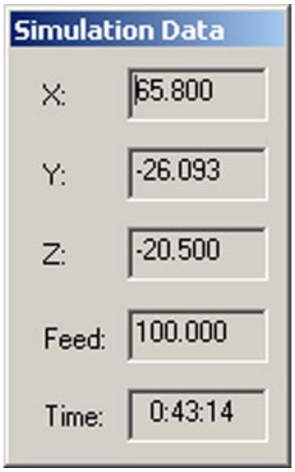
8.2.2 Simulation toolbar



The above icons in the toolbar enable you to zoom and pan the simulation picture.

8.2.3 Show Data

The check box below gives you complete information about the tool path such as coordinates of the current point, time, feed etc...



X: 65.800

Y: -26.093

Z: -20.500

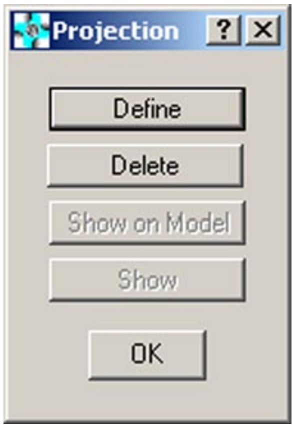
Feed: 100.000

Time: 0:43:14

- The **X**, **Y**, and **Z** fields display the position of the tool relative to the home position at every step of the simulation.
- **Feed** displays the current feed rate.
- **Time** shows the elapsed machining time of the simulated jobs (theoretical value based on feed and distance covered).

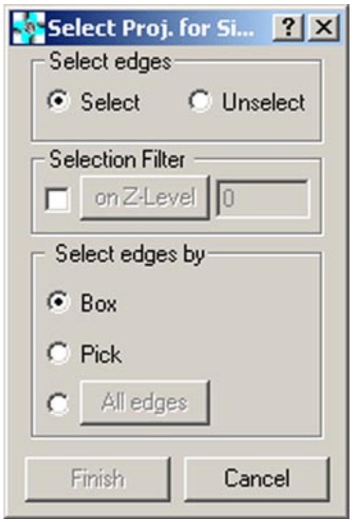
8.2.4 Projection

This option enables you to display the simulation on a Top view projection of the CAD model. When you choose this option, the following menu will be displayed:



Define

This option enables you to define entities of the CAD model that are taken into account in the Top view projection. When you choose this option, the following screen will be displayed:


Select Proj. for Si...
Select edges
● Select
○ Unselect
Selection Filter
☐ on Z-Level
0
Select edges by
● Box
○ Pick
○ All edges
Finish
Cancel

- **Select edges**

Select	This option enables you to select the edges of the CAD model either by Box, Pick or All edges.
Unselect	This option enables you to unselect edges that were previously selected.

- **Selection Filter**

This option enables you to select edges on a specific Z-level. When you choose this option you can either pick the Z-level with the cursor on the model or you can enter the Z-level in the dialog box. Once you specify the Z-level, only those edges on that level will be picked.

- **Select edges by**

Box	This option enables you to select/unselect edges by enclosing them with a box.
Pick	This option enables you to select/unselect the edges one by one.
All edges	This option automatically selects/unselects all the edges.

- **Finish**

This option enables you to build the chain after you pick the edges.

- **Cancel**

This option enables you to exit this dialog box.

Delete

This option enables you to delete what you had previously defined.

Show on model

This option enables you to see the geometry that was selected in the CAD model.

Show

This option enables you to see the model in the simulation screen.

8.2.5 Show tool

Show tool toggles on/off a graphic simulation of the tool.

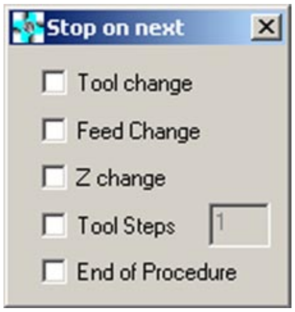
The **Parameter** field describes with what frequency SolidCAM must show the tool position.

8.2.6 Erase on Z change

Automatically clears the image after every Z change of the tool.

8.2.7 Stop on Next

This option displays the **Stop on Next dialog** where you can define the specific point to stop the simulation process.



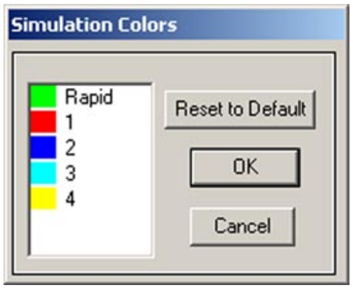
- **Tool change** - This option stops the simulation every time when the tool changes.
- **Feed change** - This option stops the simulation every time when the feed changes.
- **Z change** - This option stops the simulation at every change of the Z coordinate of the tool.
- **Tool Steps** - Stops the simulation every defined number of tool steps.
- **End of procedure** - Stops the simulation at the end of the procedure.

8.2.8 Clear

This button clears the simulated tool path and the tool images from the screen.

8.2.9 Colors

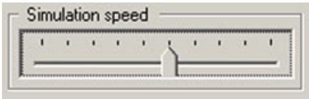
Click on the **Colors** button to change the display color of the simulated tool path. The **Simulation Colors dialog** will be displayed.


Simulation Colors
Rapid
1
2
3
4
Reset to Default
OK
Cancel

1. To change the tool color, click on the colored box left to the tool number.
2. The standard color dialog of Windows will be displayed.
3. Select the new tool path color for this tool number.
4. Confirm with **OK**.

You can use the **Reset to Default** button to apply the default tool colors, as defined in the CAM settings dialog.

8.2.10 Simulation speed

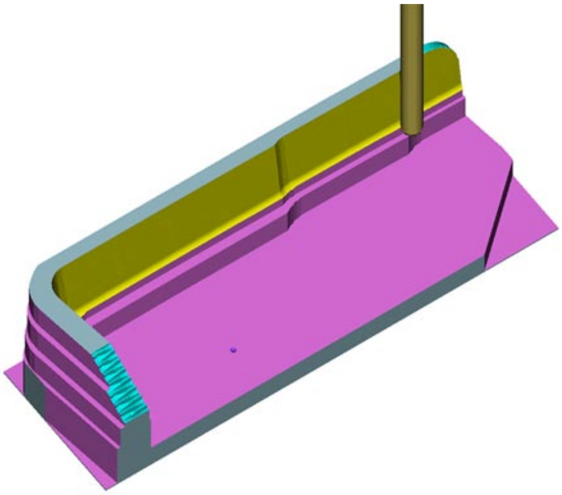

Simulation speed

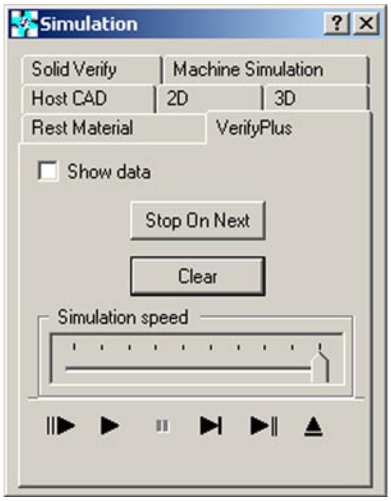
The **slider** controls the **Simulation speed**.

8.3 VerifyPlus simulation mode

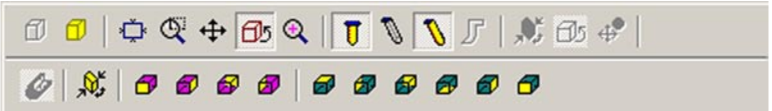
This simulation mode provides you with additional verification capabilities:

- 4/5 axes multi-sided solid simulation.
- Simultaneous 4-axes solid simulation.
- 2-D turning solid simulation.
- Turning with mill driven-tools solid simulation.




Simulation
Solid Verify
Machine Simulation
Host CAD
2D
3D
Rest Material
VerifyPlus
☐ Show data
Stop On Next
Clear
Simulation speed

8.3.1 Simulation toolbar

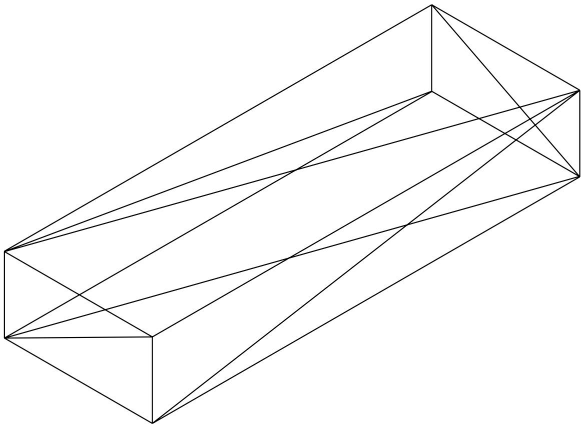


The toolbar enables you to control the simulation in this mode.

Setup and Simulation modes

VerifyPlus has two working modes: for **Setup** and for **Simulation**.

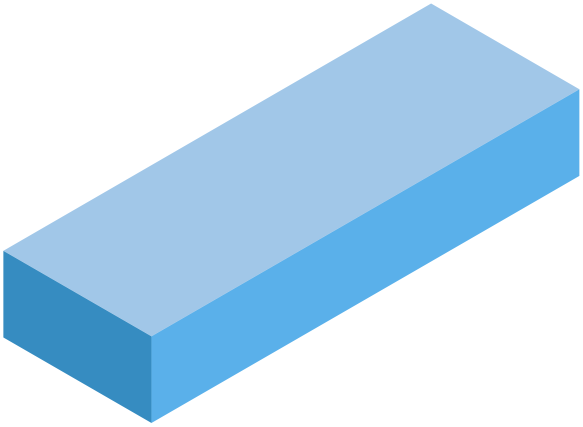
- **Setup mode**



In this mode VerifyPlus enables you to specify the scale and orientation of the model.

- **Simulation mode**

After setup, switch to the simulation mode. **VerifyPlus** shades the prepared block and prepares it for simulation.



In the Simulation mode, machining can be simulated with standard SolidCAM simulation controls. In this mode all controls for setup of the simulation (scale and orientation) are disabled. You can turn on/off the visualization of tool and also specify the tool visualization as either wireframe or shaded. You can also rotate the display and define the light controls.

8.3.2 Buttons available in the Setup Mode

Fit to window

Fits the CAM-Part to the entire screen.

Fit by box

The area you want to view in the simulation screen is defined by box.

Move

Moves the CAM-Part to any point on the screen.

Rotate

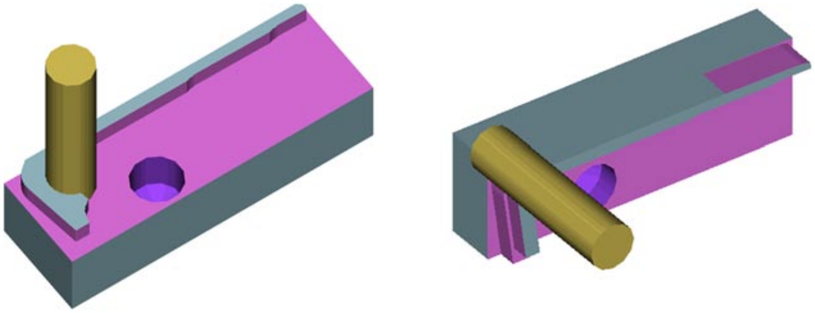
Rotates the model.

Zoom

Zooms in/out the image on the screen.

Machine orientation

This button enables you to flip the CAM-Part back and forth by 90 degrees. This option is needed to do realistic simulation on vertical/horizontal machines.



Vertical machine simulation **Horizontal machine simulation**

Isometric views buttons

These buttons enable you to view the CAM-Part in the left, right, back or front isometric views.

View buttons

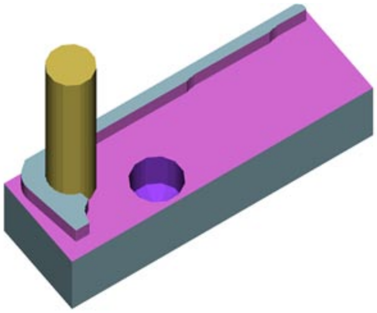
These buttons enable you to view the CAM-Part in the top, left, back, bottom, right or front views.

Show/Hide Tool button

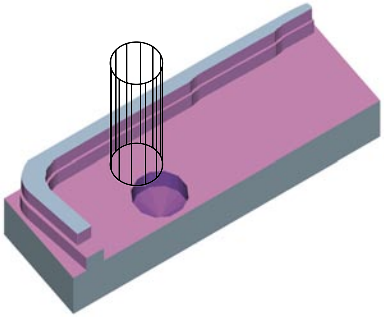
Pressing this button enables you to show or hide the tool during simulation. This option is also available in the simulation mode.

Wire Tool/Shaded Tool

These buttons enable you to see the tool as a wire frame tool or as a solid tool.



Shaded tool simulation



Wireframe tool simulation

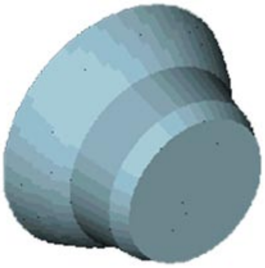
The wireframe tool will always be seen in it entirety, whereas the solid tool is partially hidden by the CAM-Part.

Show/Hide Tool Holder.

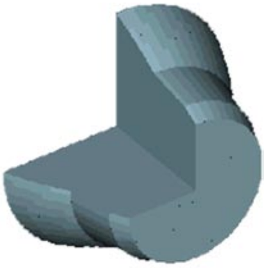
This button is active in turning only. When pressed, it allows you to see the holder of the turning tool.

Section View

This button is active in Turning only. When pressed, it allows you to see the whole CAM-Part or with a quarter cut away.



Section view mode



Normal mode

8.3.3 Buttons available in the Simulation Mode

Flip

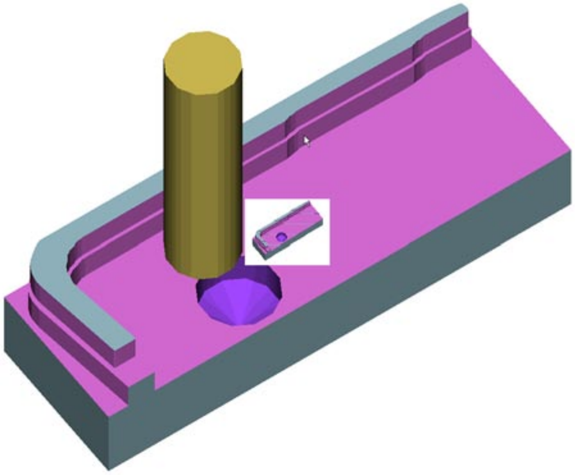
This button flips the CAM-Part back and forth. We need it in the simulation mode to see other sides of the CAM-Part during the simulation. This option is useful for multi-sided simulation.

Solid Rotate

This button enables you to rotate the CAM-Part to any position after simulation without exiting to Setup mode.

To rotate the solid, click on any location inside the Graphics view using the left button of the mouse, hold the button down and drag the cursor across the screen. A small inset window at the center of the view will display a continuous snapshot of the current orientation of the model. When you release the button, the entire view is updated to show the new orientation of the model.

The effect of the change in rotation is not permanent, and this change has to be reset to proceed with the Simulation.



Light control

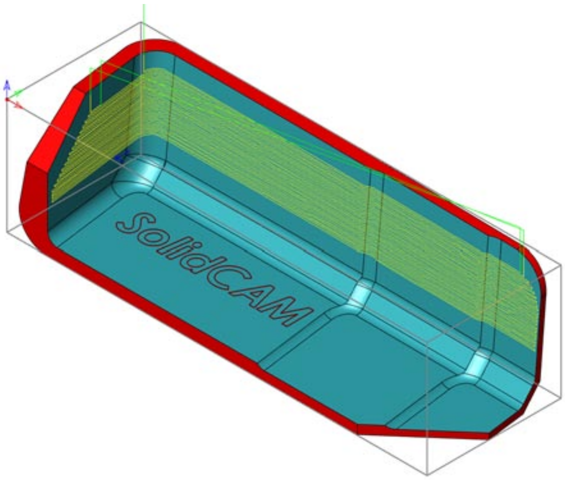
This command enables you to use the left button of the mouse to change the light source on the solid model in the Simulation mode.

To change the light source, click on any location inside the Graphics view using the left button of the mouse, hold the button down and drag the cursor around within the bounds of the view. A small inset window at the center of the view will display a snapshot of the current lighting effect on the model. When you release the button, the entire view is updated to show the light change effect on the complete model.

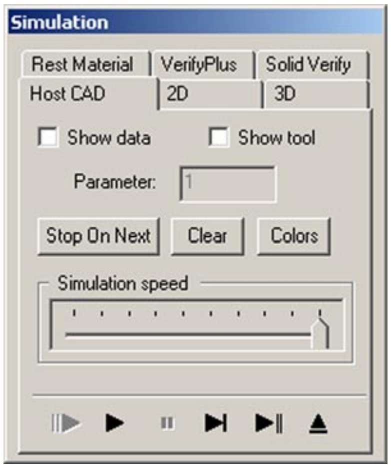
The Move light source command is used for visual purposes only. The effect of the change in light source direction is not permanent. The light source change has to be reset before proceeding with the Simulation.

8.4 Host CAD simulation mode

This simulation mode enables you to display the tool path directly on the model in the Host CAD window. Since all the View options of the Host CAD are active during the simulation, you can see the tool path from different perspectives and zoom in on a certain area of the model.



8.4.1 Simulation Control


Simulation
Rest Material
VerifyPlus
Solid Verify
Host CAD
2D
3D
☐ Show data
☐ Show tool
Parameter: 1
Stop On Next
Clear
Colors
Simulation speed

8.4.2 Show Data

This check box gives you information about the tool path such as coordinates of the current point, time, feed etc...

For more information on the **Show Data** option, please refer to the topic **8.2.3**.

8.4.3 Show Tool

- **Show tool** toggles on/off a graphic simulation of the tool.

The **Parameter** field describes with what frequency SolidCAM must show the tool position.

8.4.4 Stop on Next

This option opens the **Stop on Next dialog** where you can define the specific point to stop the simulation process.

For more information on the **Stop on Next** option, please refer to the topic **8.2.7**.

8.4.5 Clear

This button clears the simulated tool path and the tool images from the Host CAD screen.

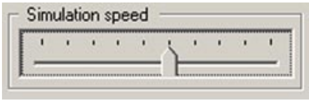
8.4.6 Colors

Click on the **Colors** button to change the display color of the simulated tool path. The **Simulation Colors dialog** will be displayed.

For more information on the **Colors** option, please refer to the topic **8.2.9**.

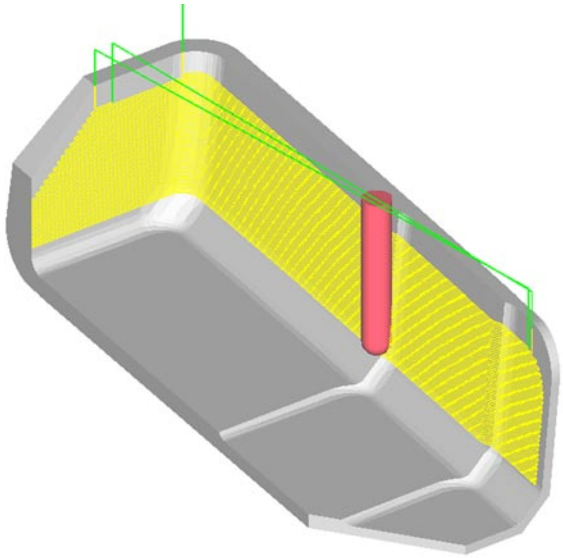
8.4.7 Simulation speed

You can use the **slider** to control the **Simulation speed**.



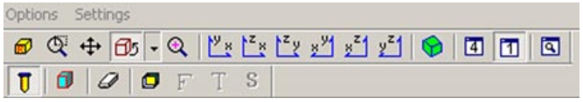
8.5 3D simulation mode

This simulation mode enables you to display the tool path on the 3D model using all the graphic acceleration functionality of OpenGL. The **target model** must be defined to use this simulation mode. For more information on the **Target model**, please refer to the topic **2.4**.



8.5.1 Simulation toolbar

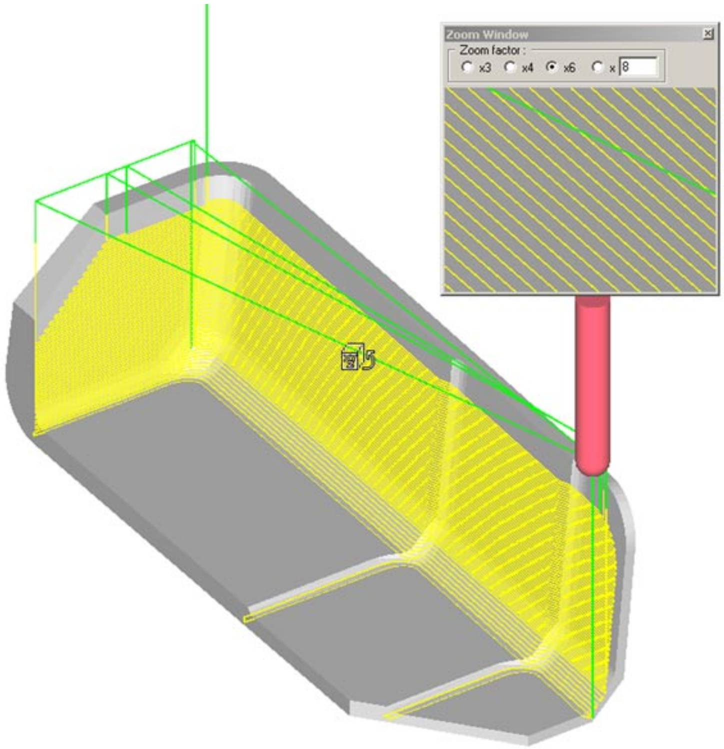
SolidCAM enables you to use the toolbar to control the simulation in this mode.



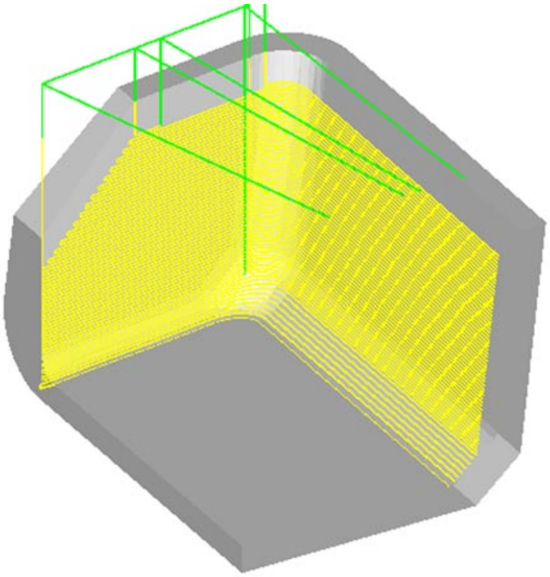
The following icons are available in this toolbar:

- These icons are used for model scaling, panning and rotating.
- This icon enables you to remove the simulated tool path from the screen.
- With these icons you can toggle on/off the tool visualization during the simulation.
- This icon enables you to switch the graphic screen between three modes:
    - Tool path only simulation
    - Tool path and model simulation
    - Model only simulation
- These icons enable you to show the model in the projection views (Left, Right, Top etc...)

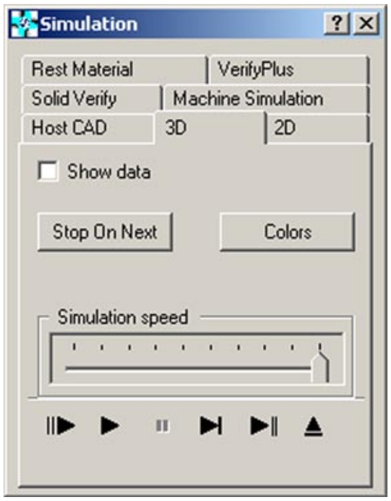
- This icon returns the original position and orientation of the model.
- These icons switch the simulation to 4/1 view mode.
- This option enables you to show the Zoom Window with the enlarged image of the model and tool path fragment centered on the mouse position. You can control the zoom factor in the Zoom Window.



- These icons enable you to show the dynamic section of the model. The section slider will be displayed in the graphic area. By moving this slider you can control the section.



8.5.2 Simulation control


Simulation
Rest Material
VerifyPlus
Solid Verify
Machine Simulation
Host CAD
3D
2D
☐ Show data
Stop On Next
Colors
Simulation speed

8.5.3 Show Data

This check box gives you information about the tool path such as coordinates of the current point, time, feed etc...

For more information on the **Show Data** option, please refer to the topic **8.2.3**.

8.5.4 Stop on Next

This option opens the **Stop on Next dialog** where you can define the specific point to stop the simulation process.

For more information on the **Stop on Next** option, please refer to the topic **8.2.7**.

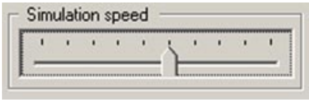
8.5.5 Colors

Click on the **Colors** button to change the display color of the simulated tool path. The **Simulation Colors dialog** will be displayed.

For more information on the **Colors** option, please refer to the topic **8.2.9**.

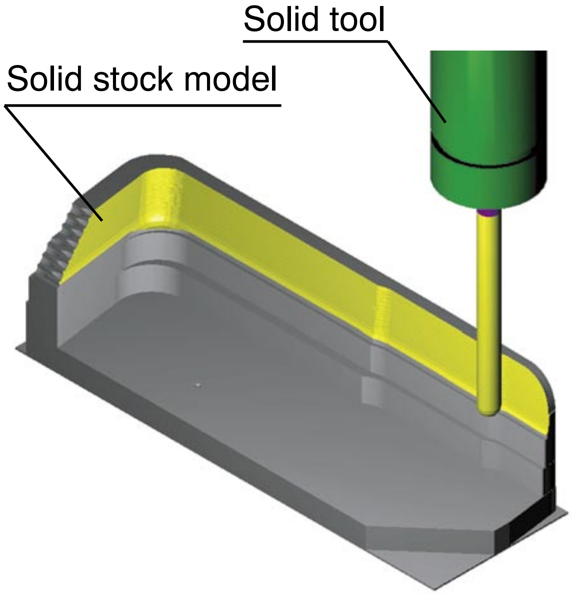
8.5.6 Simulation speed.

You can use the **slider** to control the **Simulation speed**.

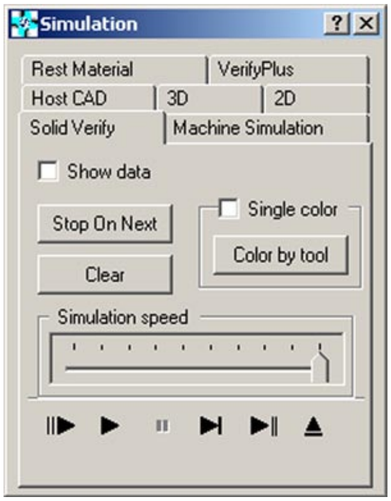

Simulation speed

8.6 SolidVerify simulation mode

This mode enables machining simulation on the solid model. The solid stock model (defined by **material boundaries**) is used in this mode. During the machining simulation process, SolidCAM subtracts the tool movements (using solid Boolean operations) from the solid model of the stock. The remaining machined stock is a solid model that can be dynamically zoomed or rotated.


Solid tool
Solid stock model

8.6.1 Simulation Control


Simulation
Rest Material
VerifyPlus
Host CAD
3D
2D
Solid Verify
Machine Simulation
☐ Show data
Stop On Next
Clear
☐ Single color
Color by tool
Simulation speed

8.6.2 Show Data

This check box offers you complete information about the tool path such as coordinates of the current point, time, feed etc...

For more information on the **Show Data** option, please refer to the topic **8.2.3**.

8.6.3 Stop on Next

This option opens the **Stop on Next dialog** where you can define the specific point to stop the simulation process.

For more information on the **Stop on Next** option, please refer to the topic **8.2.7**.

8.6.4 Clear

This option enables you to clear the simulation and return to the unmachined stock.

8.6.5 Simulation speed

SolidCAM enables you to use the **slider** to control the **Simulation speed**.

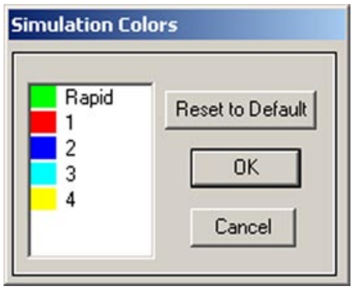
8.6.6 Single color/Color by tool

- **Single color**

  This option enables you to use a single color for all tools during the simulation process.

- **Color by tool**

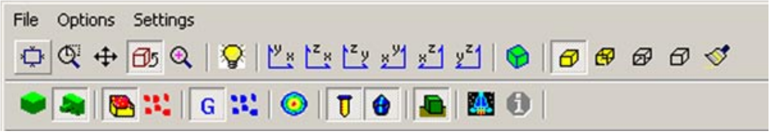
  Use this option to define a specific color for each tool. Click on the **Color by tool** button to set the colors. The **Simulation Colors dialog** will be displayed.



1. To change the tool color, click on the colored box left to the tool number.
2. The standard color dialog of Windows will be displayed.
3. Select the new tool path color for this tool number.
4. Confirm with **OK**.

You can use the **Reset to Default** button to apply the default tool colors, as defined in the CAM settings dialog.

8.6.7 Simulation toolbar



The toolbar below enables you to control the simulation in this mode.

Available buttons:

Fit to window

Fits the CAM-Part to the entire screen.

Fit by box

The area you want to view in the simulation screen is defined by a box.

Move

Moves the CAM-Part to any point on the screen.

Rotate

Rotates the model.

Zoom

Zooms in/out the image on the screen.

Light control

This command enables you to use the left button of the mouse to change the light source on the solid model in the Simulation mode.

To change the light source, click on any location inside the Graphics view using the left button of the mouse, hold the button down and drag the cursor around within the bounds of the view.

Isometric views buttons

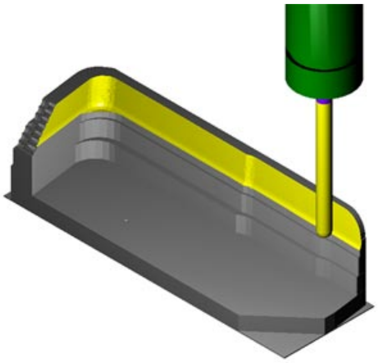
These buttons enable you to view the CAM-Part in the left, right, back or front isometric views.

View buttons

These buttons enable you to view the CAM-Part in the top, left, back, bottom, right or front views.

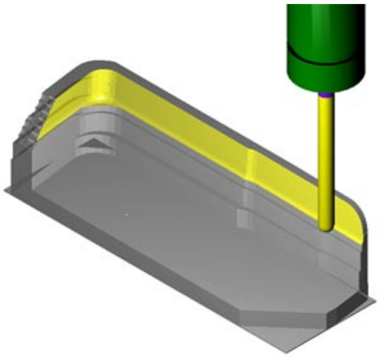
Render mode

Displays a shaded view of the model.



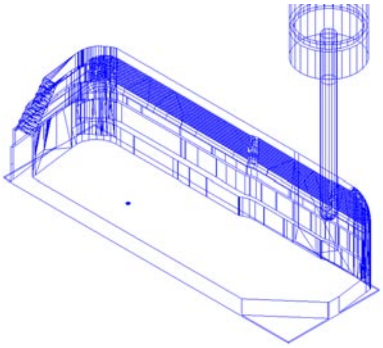
Hybrid mode

Displays a transparent view of the model.



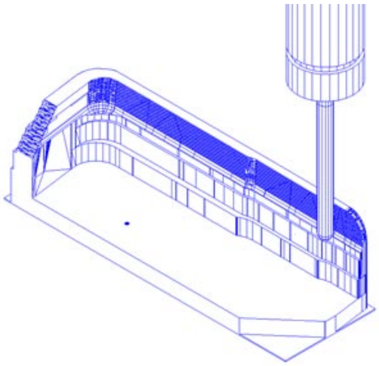
Wire mode

Displays the CAM-Part and tool movements in wireframe mode.



Hidden line mode

Displays the CAM-Part and the tool movements in hidden-line wireframe mode.



Redraw

Redraws the view.

Show/Hide Machined Stock

Shows/hides the machined stock model

Show/Hide Target

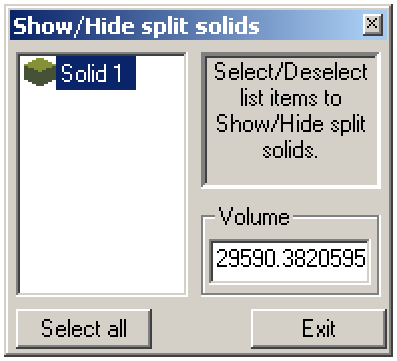
Shows/hides the target model.

Show/Hide Rest Material

Shows/hides the rest material on the machined stock model.

Show/Hide Split Solids

This option displays the **Show/Hide Split solids dialog**.



Select/Deselect list items to Show/Hide split solids.

Volume: 29590.3820595

This dialog enables you to switch between split solids during the simulation and enables you to remove the pieces of material that have been cut away during the machining

Show/Hide Gouges

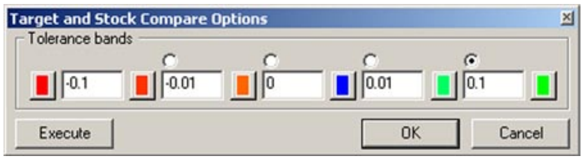
Shows gouges of the target model.

Split Gouges

This option displays the **Show/Hide Split solids dialog**. This dialog enables you to switch between split solids of gouge areas during the simulation and enables you to check separately the areas of the gouge

Compare target and machined stock

This option enables you to compare the stock model and target model and assign different colors to different rest material areas, depending on their tolerance.



Target and Stock Compare Options

Tolerance bands

-0.1 ○ -0.01 ○ 0 ○ 0.01 ● 0.1

1. Set the tolerance values and colors in the **Target and Stock Compare Options dialog**.
2. Press **Execute** to show the comparison.
3. Press **OK** to exit.

Activate/Disactivate initialization of machined stock

This option is used in the case of single job simulation.

If this option is turned on, SolidCAM enables you to perform the simulation on the stock machined by previous jobs.

If this option is turned off, SolidCAM performs the simulation on the original unmachined stock model.

Show/Hide Tool

Shows or hides the tool during simulation.

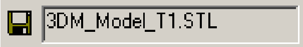
Show/Hide Holder

Shows or hides the tool holder during simulation.

Show/Hide Fixture

Shows or hides the fixture during simulation.

For more information on the **Fixtures** definition, please refer to the topic **5.6**.



3DM_Model_T1.STL

Save the simulation model

This field enables you to save the current model of the material removal in STL format.

This field is active only when the **Solid Rest material calculation method** and **Manual method of the Rest material update** are chosen.

For more information on this subject, please refer to the topic **2.4**.

3DM_Model_T1

Load the simulation model

This field enables you to load the previously saved model of the material removal. The simulation will be performed on the loaded model.

This field is active only when the **Solid Rest material calculation method** and **Manual method of the Rest material update** are chosen.

For more information on this subject, please refer to the topic **2.4**.

8.6.8 Simulation menu

File
Options
Settings

File

- **Open**

  Open machined stock from FCT

  This option enables you to load the machined stock model from the SolidCAM FCT file (*.fct).

  Open machined stock from STL

  This option enables you to load the machined stock model from the Stereolithography file (*.stl).

- **Save machined stock to STL**

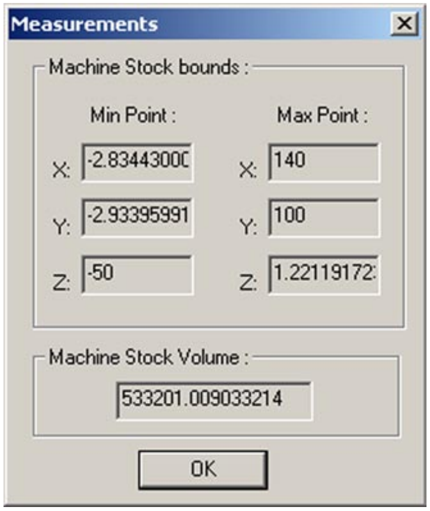
  This option enables you to save the machined stock model to the Stereolithography file (*.stl).

Options

- **Compare target and machined stock**

This option explained above enables you to compare the stock model and target model and assign different colors to different rest material areas, depending on their tolerance.

This option displays the **Measurements dialog** where you can see the dimensions and the volume of the Machined stock model.


Measurements
OK

Machine Stock bounds :

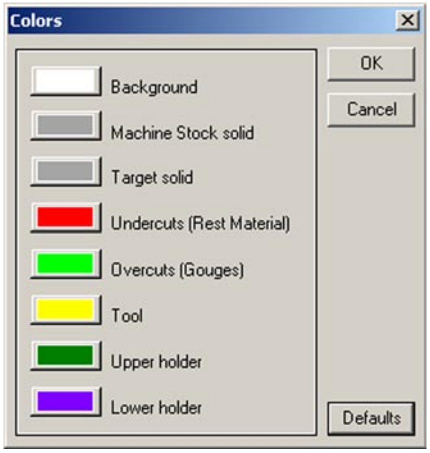
	Min Point :		Max Point :
X:	-2.8344300	X:	140
Y:	-2.93395991	Y:	100
Z:	-50	Z:	1.22119172

Machine Stock Volume : 533201.009033214

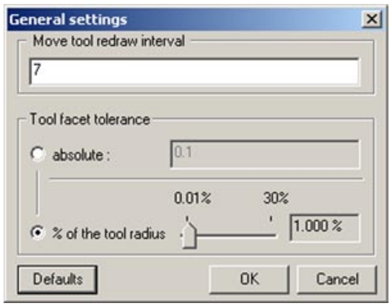
Settings

- **Colors**

This option enables you to choose colors for the simulation. The **Colors dialog** will be displayed.


Colors
OK
Cancel
Background
Machine Stock solid
Target solid
Undercuts (Rest Material)
Overcuts (Gouges)
Tool
Upper holder
Lower holder
Defaults

- **General**



Move tool redraw interval: 7

Tool facet tolerance:

○ absolute : 0.1

● % of the tool radius: 1.000 %

Move tool redraw interval.

This option enables you to expedite the simulation by defining the number of tool movements until the next redraw of the display. The default value is **7**.

Tool facet tolerance.

This controls the precision of the tool for the simulation.

- **Absolute**

  This parameter defines the precision of the facets of a tool in mm (or inches). Since this is an absolute value, the facets tolerance for the small and large tools will be the same. During the simulation, SolidCAM produces the Boolean subtraction of the tool from the CAM-Part. This tolerance will influence the quality of the simulation. The quality of the simulation will be the same for large tools and small tools. The speed of the simulation will be relative to the tool. The simulation of the lager tool will be slower.

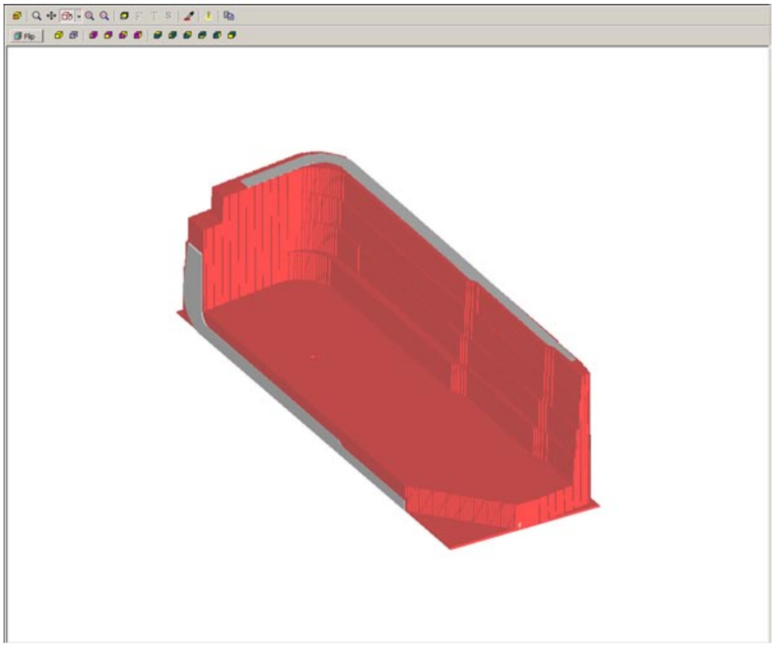
- **% of the tool radius**

  This parameter defines the precision of the facets of a tool in percentages to the tool radius. In this case the facet size will differ from tool to tool. However the amount of facets on the tools will be the same. The quality of the simulation will be relative to the tool. The speed of the simulation will be the same in small tools and large tools.

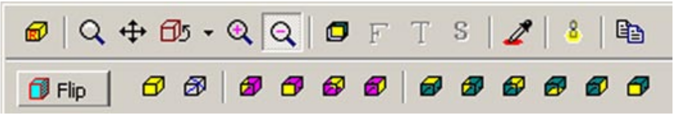
8.7 Rest Material simulation mode

This simulation mode enables you to display the model with the rest material before and after each Job. It also displays existing gouge areas so that it can be prevented in the real machining. To use this simulation mode, the **Target model** must be defined and the **Update stock model** option must be active. For more information about the Target Model and the Update stock model option, please refer to the topic **2.4**

This mode will be used only when the Z Buffer mode will be chosen for the Rest Material calculation. When the Solid mode is chosen, the Rest Material simulation will be performed by SolidVerify.

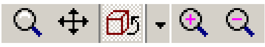


8.7.1 Simulation toolbar



SolidCAM enables you to control the simulation in this mode with the toolbar.

Available buttons:

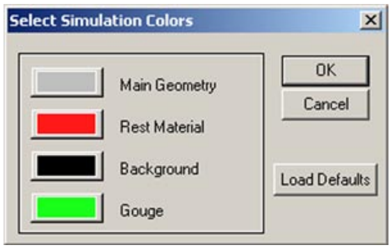


Model orientation buttons

This set of icons is for model scaling, panning and rotating.

Colors

This button enables you to define colors for the simulation. The **Select Simulation colors dialog** will be shown.



Click on the color button to define the color.

Lighting

This icon enables you to control the lighting of the rest material.

Copy

This icon enables you to copy the simulation image into the Windows clipboard for future use.

Shaded/wireframe mode

These buttons enable you to switch to the shaded/wireframe mode.

Isometric view buttons

These icons enable you to rotate the model into isometric views (Left, Right, etc...)

View buttons

These buttons enable you to view the CAM-Part in the top, left, back, bottom, right or front views..

Refresh

This icon returns the original position and orientation of the model.

Flip

Flip

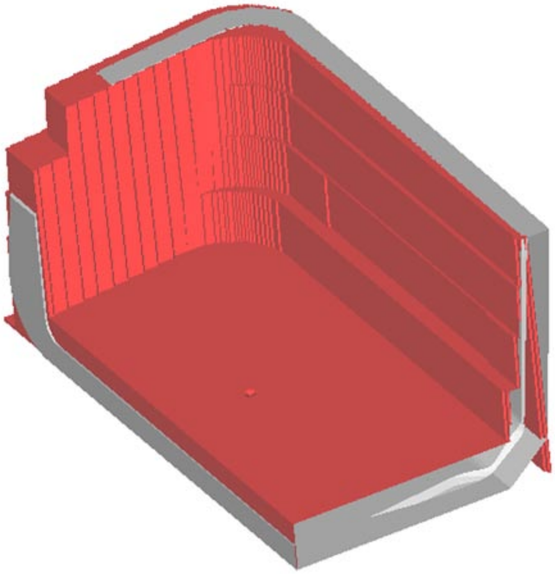
This button switches between the following simulation modes:

- Model and rest material
- Rest material only
- Model only

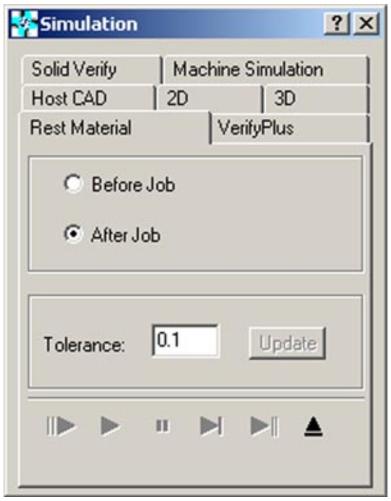
F T S

Dynamic section

These icons enable you to show the dynamic section of the model. The section slider will be displayed in the graphic area. By moving this slider you can control the section.



8.7.2 Simulation control


Simulation
Solid Verify
Machine Simulation
Host CAD
2D
3D
Rest Material
VerifyPlus
Update

○ Before Job

● After Job

Tolerance: 0.1

With the **Before Job** and **After Job** options you can display the rest material either before or after the machining within the selected Job or the whole CAM-Part.

8.7.3 Tolerance

With this parameter you can control the precision of the rest material simulation. The smaller the tolerance parameter, the better the precision.

1. Set the tolerance value.
2. Click on the update button to redraw the rest material picture according to the new tolerance.

G-Code

9

This command enables you to generate and display the G-Code file for the CAM-Part or for machining jobs. The G-Code file can then be transferred to your CNC-machine using a floppy disk, via a DNC (RS 232) or Ethernet connection.

The format of the NC-program depends on the Controller you selected in the CAM-Part data dialog. The G-Code output format can be customized using the GPP interface or by changing the controller's *.mac and *.gpp files.

❐ Generating G-Code

This command enables you to generate and display the G-Code file for the CAM-Part or for machining jobs. The G-Code file can then be transferred to your CNC-machine using a floppy disk, via a DNC (RS 232) or Ethernet connection.

The format of the G-Code depends on the Controller you selected in the CAM-Part data dialog. The G-Code output format can be customized either by using the GPP interface or by changing the controller‘s *.mac and *.gpp files. Contact **technical support** for more information on post-processor customization.

9.1 Generate

This option generates all the G-Code files for the CAM-Part after which the file will be displayed using the text editor defined in **Settings>Editors**.

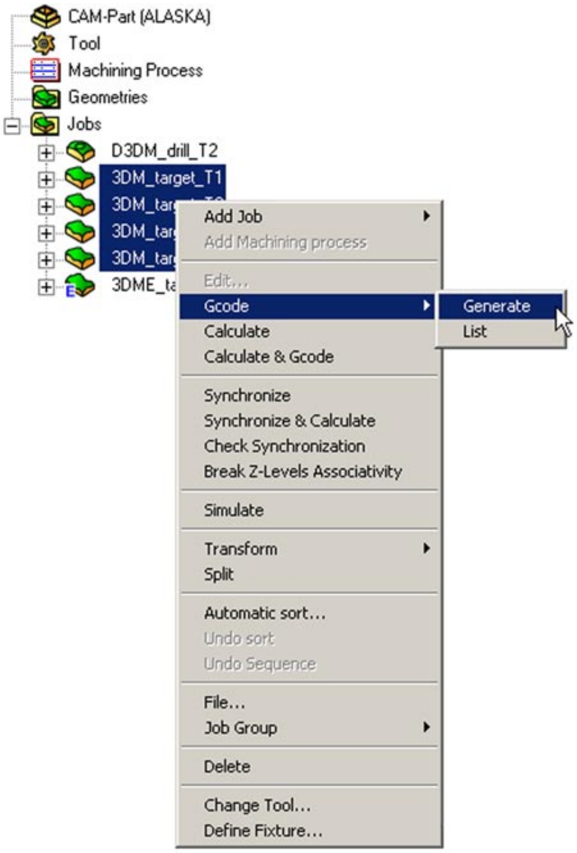
If any changes were made in the job or in the *.mac or the *.gpp post-processor files, you have to generate the G-Code again.

- **G-Code All**

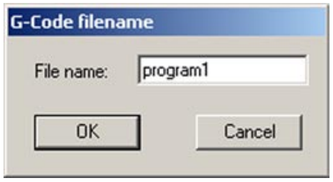
  This command is available when you press the right mouse button on the job symbol. You will receive one G-Code file including the necessary tool changes, unless you have separated jobs or job groups using **split marks**. The G-Code file will be saved to the CAM-Part‘s directory and will carry the name: partname.tap.

- **G-Code for a single job or for a group of jobs**

1. Click on the Job you want to generate the G-Code for or select **a group of jobs** using the mouse in combination with the **Shift**/**Ctrl**-key. Then press the right mouse button and choose Generate from the G-Code menu.


CAM-Part (ALASKA)
Tool
Machining Process
Geometries
Jobs
D3DM_drill_T2
3DM_target_T1
3DM_tar
3DM_tar
3DM_tar
3DME_ta
Add Job
Add Machining process
Edit...
Gcode
Calculate
Calculate & Gcode
Synchronize
Synchronize & Calculate
Check Synchronization
Break Z-Levels Associativity
Simulate
Transform
Split
Automatic sort...
Undo sort
Undo Sequence
File...
Job Group
Delete
Change Tool...
Define Fixture...
Generate
List

2. The **G-Code filename dialog** prompts you to enter a name for the *.tap file.

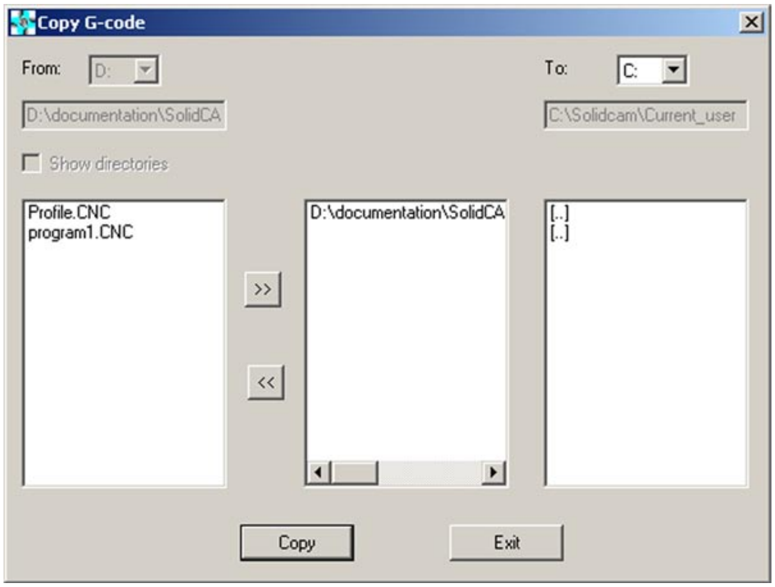

G-Code filename
OK
Cancel

File name: program1

9.2 List

This option enables you to display a list of the already generated G-Code files for the current CAM-Part or job. The G-Code (*.tap) file will be opened with the text editor defined in **Settings>Editors**. Use the editor to change, save or print the G-Code file.

9.3 Copy



From: D:

D:\documentation\SolidCA

To: C:

C:\Solidcam\Current_user

Profile.CNC
program1.CNC

D:\documentation\SolidCA

[..]
[..]

This option enables you to copy G-Code files. When you choose this option, the list of the generated G-Code files for the current CAM-Part is displayed in the **Copy G-Code dialog**.

1. Choose the ***.tap** file you want to copy to another directory (for multiple selections use the **Ctrl** and **Shift**-key).
2. Transfer the list of files you want to copy to the middle field by pressing the Left to Right arrow.
3. Choose the drive and the directory you want to copy to.
4. Press the **Copy** field in order to execute the **Copy** command.

9.4 Print

This option enables you to print the G-Code file.

SolidCAM Settings

10

The SolidCAM Settings feature enables you to define a set of SolidCAM defaults such as SolidCAM folders, units, tolerances, CNC controllers etc...

In the settings dialog you can change the path to related applications, units, tolerances, default machines, editors, and color settings for SolidCAM. The Settings register enables you to easily view and edit the current program preferences and options.

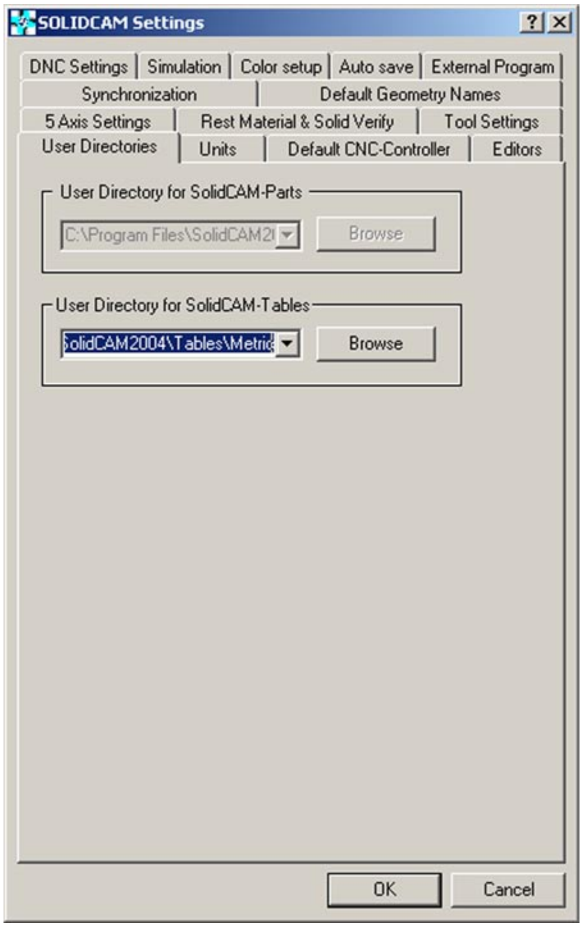
❒ User Directories Settings
❒ Units Settings
❒ Simulation Settings
❒ Synchronization settings
❒ Rest Materal & SolidVerify Settings
❒ Tool Settings

10.1 Introduction

In the Settings dialog you can change the path to related applications, units, tolerances, default machines, editors, and color settings for SolidCAM.

The **Settings** register enables you to easily view and edit the current program preferences and options.

- **DNC Settings**
- **Simulation settings**
- **Color Setup**
- **Auto-save settings**
- **External Program settings**
- **Synchronization settings**
- **Default Geometry Names**
- **Rest Materal & SolidVerify**
- **Tool Settings**
- **User Directories settings**
- **Units Settings**
- **Default CNC-Controller**
- **Editors settings**

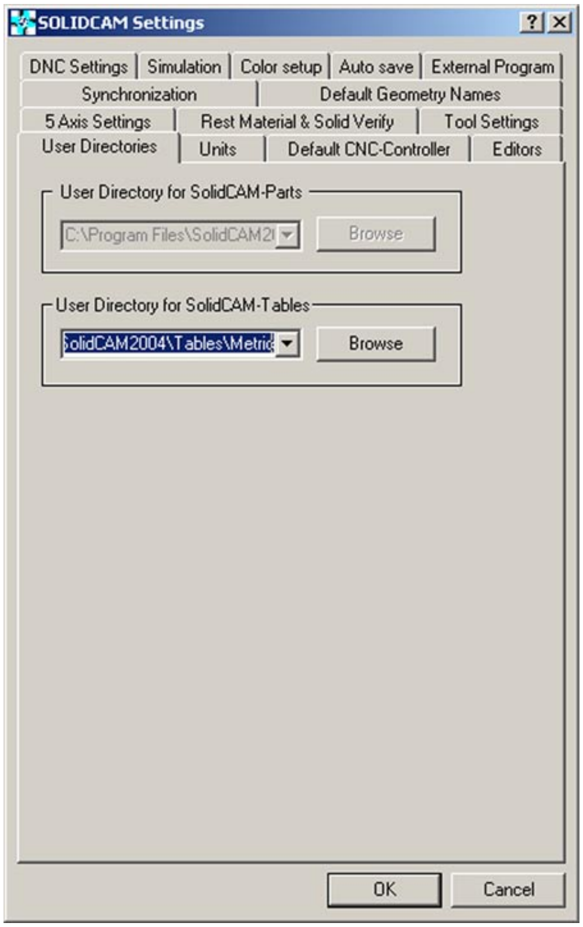

SOLIDCAM Settings
DNC Settings | Simulation | Color setup | Auto save | External Program
Synchronization | Default Geometry Names
5 Axis Settings | Rest Material & Solid Verify | Tool Settings
User Directories | Units | Default CNC-Controller | Editors
Browse
Browse
OK
Cancel

User Directory for SolidCAM-Parts: C:\Program Files\SolidCAM2

User Directory for SolidCAM-Tables: olidCAM2004\Tables\Metric

10.2 User Directories Settings

The **Path** register enables you to set the path to your post-processor files and to your user-directory.


SOLIDCAM Settings
DNC Settings
Simulation
Color setup
Auto save
External Program
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
Browse
Browse
OK
Cancel

User Directory for SolidCAM-Parts: C:\Program Files\SolidCAM2

User Directory for SolidCAM-Tables: olidCAM2004\Tables\Metric

10.2.1 User Directory for SolidCAM-Parts

The user directory defines the default folder in which CAM-Parts and G-Code files created with SolidCAM are stored. Whenever you load a CAM-Part, SolidCAM will browse the user directory for SolidCAM-Part files (***.prt**). All newly created CAM-Part files will be placed in the user directory by default.

1. Click on the **Browse** button.
2. The browser window will be loaded.
3. Choose the folder you want from the browser.

10.2.2 User Directory for SolidCAM-Tables

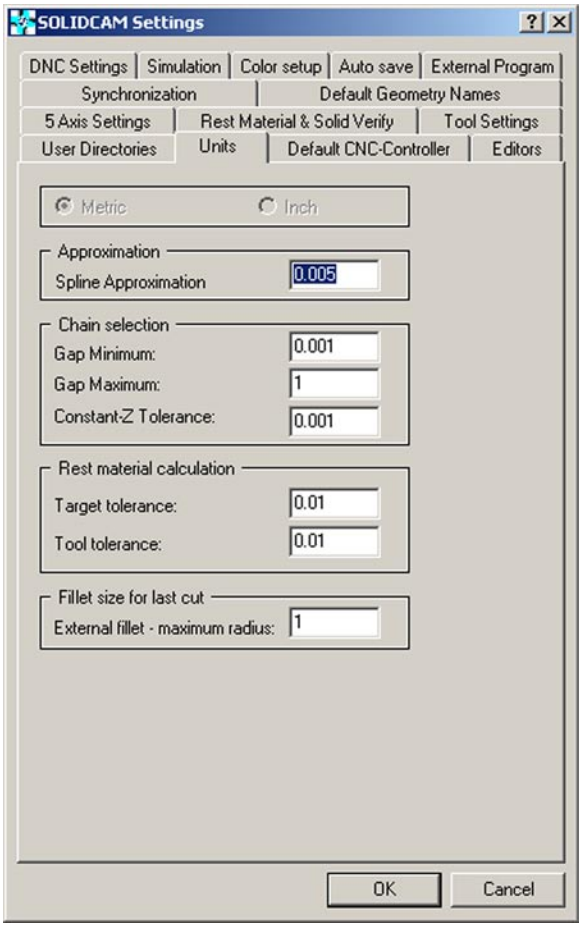
SolidCAM uses this folder to store **Tool Tables** and **Machining processes**.

- The ***.tab** file in the SolidCAM Tool Table file contains detailed information about tools.
- The ***.mpt** file is the Machining Process Table file. In addition to this file, SolidCAM creates the Machining Process Folder with a set of SolidCAM‘s auxiliary files.

1. Click on the **Browse** button.
2. The browser window will be loaded.
3. Choose the folder you want from the browser.

10.3 Units Settings

The **Units** register enables you to set the unit system, spline approximation and chain tolerances.


SOLIDCAM Settings
DNC Settings
Simulation
Color setup
Auto save
External Program
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
OK
Cancel

● Metric ○ Inch

Approximation

Spline Approximation: 0.005

Chain selection

Gap Minimum: 0.001
Gap Maximum: 1
Constant-Z Tolerance: 0.001

Rest material calculation

Target tolerance: 0.01
Tool tolerance: 0.01

Fillet size for last cut

External fillet - maximum radius: 1

10.3.1 Metric/Inch

This option enables you to select the unit system you want to work with.

10.3.2 Approximation

Whenever you use a spline edge (NURBS) to select a chain geometry for profiles, working areas, sections etc., the spline will be transformed to lines. „Approximation on splines" defines the tolerance that is used when the spline is transformed to lines.

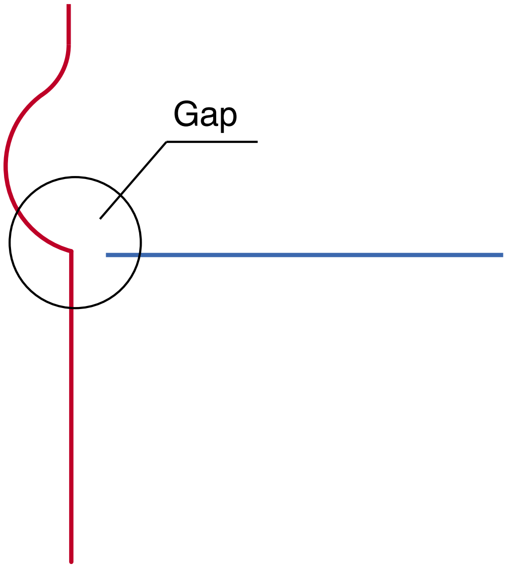
Example:

When you import an IGES surface, the surface boundaries will usually be defined with spline curves. If you mill the outside contour of the surface with a profile job, you have to select the boundary as the profile geometry. You will receive straight edges and facets on the contour if you do not adjust the spline approximation to your requirements. If you select a tighter tolerance, the G-Code file will show more points on the contour and produce a smoother edge.

This tolerance does not affect jobs that work on 2D contours such as arcs, lines or circles.

10.3.3 Chain selection

The Auto select function for selecting chain geometries uses certain tolerances to decide whether two consequent elements are connected or independent. The behavior of this function depends on the **Gap** between the entities.



Gap minimum and Gap maximum tolerance

The Auto select can function in three different ways:

- If the gap between the edges is less than the **minimum chain gap**, the system automatically continues the chain and closes the gap.
- If the gap is between the minimum and maximum values, you are asked whether the gap should be closed. If you answer **YES**, the gap is automatically closed.
- If the gap is larger than the maximum value for a chain gap, the message, **Chain must be continuous**, is displayed.

Constant-Z Tolerance

When you work in the **Auto Constant Z** mode, all connected entities on the same Z-level of the current coordinate system will be selected. This tolerance defines the maximum deviation from the Z plane where an element will still be recognized as being on the same Z-level.

It mainly depends on the size of your geometries and the accuracy of your models or imported 2D/3D files which gap and constant-Z tolerance you should use. The default setting can be used for most of the common models, but when you work with very intricate CAM-Parts, for example, a large max. gap tolerance can cause problems as you might close contours in places that should normally be not connected. To avoid this, answer „**No**“ to the message „**Do you want to close the gap?**“ and switch to single curve mode to make sure you selected the correct elements.

10.3.4 Rest Material Calculation

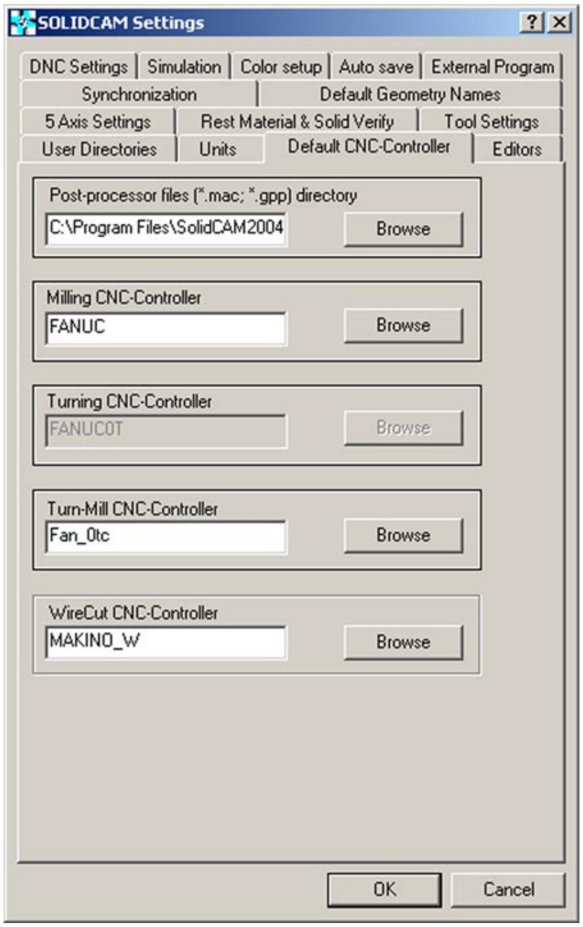
The **Target tolerance** parameter controls the accuracy of the target and stock model triangulation.

10.3.5 Fillet size for last cut

This parameter defines the maximum allowed value of the radius for the external fillet.

For more information on the **Fillet size for last cut** option, please refer to the topic **6.1.4**.

10.4 Default CNC-controller settings


SOLIDCAM Settings
DNC Settings
Simulation
Color setup
Auto save
External Program
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
Browse
Browse
Browse
Browse
Browse
OK
Cancel

Post-processor files (*.mac; *.gpp) directory: C:\Program Files\SolidCAM2004

Milling CNC-Controller: FANUC

Turning CNC-Controller: FANUC0T

Turn-Mill CNC-Controller: Fan_0tc

WireCut CNC-Controller: MAKINO_W

10.4.1 Post-processor files directory

GPP files with the extensions ***.mac** and ***.gpp** are post-processor files used by SolidCAM.

- The ***.mac** file contains information about the CNC-machine, e.g. number of simultaneous axes, available cycles, default tool positioning, etc..
- The ***.gpp** file translates the calculated tool path into G-Code. The G-Code format, cycle definitions, etc. of your G-Code file is controlled through this file.

You have to specify the path to the folder where you keep the ***.mac** and ***.gpp** files you have received from SolidCAM. The post-processors that come with the evaluation or demo version are installed in the SolidCAM program directory.

1. Click on the **Browse** button.
2. The browser window will be loaded.
3. Choose the folder you want from the browser.

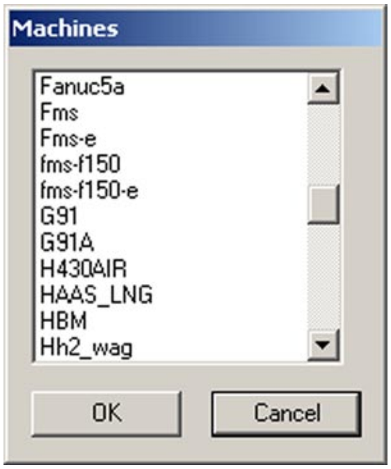
10.4.2 CNC-controllers

SolidCAM enables you to define the default machine controllers:

- **Milling CNC-Controller**
- **Turning CNC-Controller**
- **Milling&Turning CNC-Controller**
- **WireCut CNC-Controller**

The post-processors you selected will be used as the default machine, which means it will appear in the **Part Data dialog** as the controller for the new CAM-Part. Naturally, you can change the controller for the CAM-Part with the list field if you want to use another machine controller.

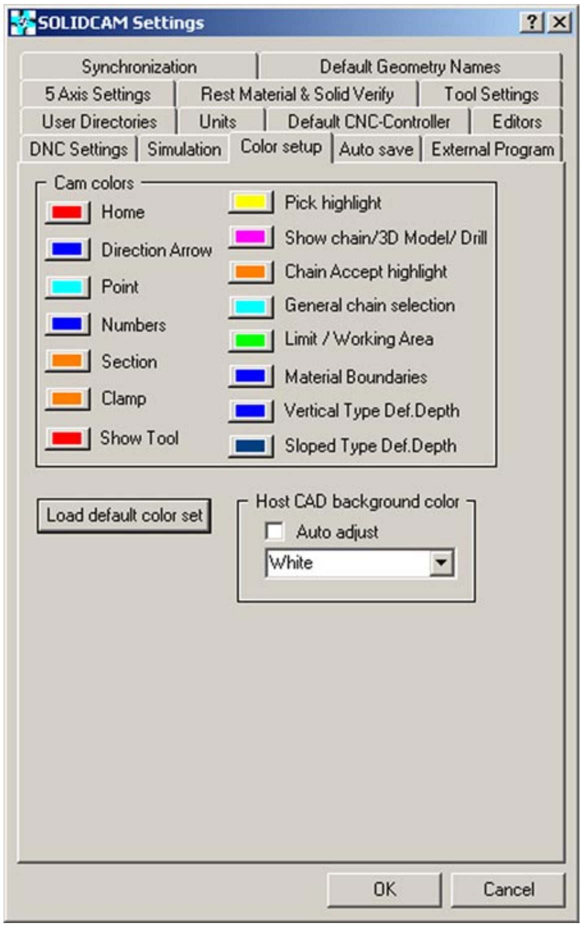
1. Click on the **Browse** button. A list of milling/turning post-processors available in your **Post-processor files directory** will be displayed in the **Machines list**.


Machines
Fanuc5a
Fms
Fms-e
fms-f150
fms-f150-e
G91
G91A
H430AIR
HAAS_LNG
HBM
Hh2_wag
OK
Cancel

2. Double click on the machine you want to use as your default machine.

10.5 Color Setup

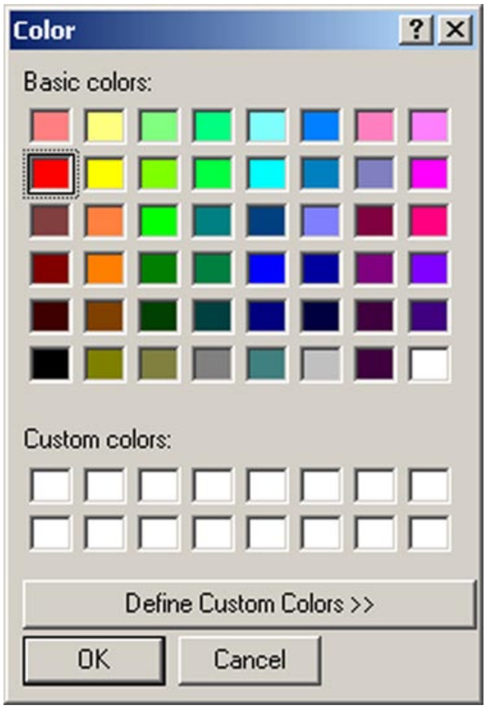
The **Color Setup** register enables you to customize the colors used by SolidCAM.


SOLIDCAM Settings
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
DNC Settings
Simulation
Color setup
Auto save
External Program
Cam colors
Home
Direction Arrow
Point
Numbers
Section
Clamp
Show Tool
Pick highlight
Show chain/3D Model/ Drill
Chain Accept highlight
General chain selection
Limit / Working Area
Material Boundaries
Vertical Type Def.Depth
Sloped Type Def.Depth
Load default color set
Host CAD background color
OK
Cancel

☐ Auto adjust

White

To change a color, click on the corresponding color button. The standard Windows **Color selection** dialog will be displayed, enabling you to pick the color you want.


Color
Basic colors:
Custom colors:
Define Custom Colors >>
OK
Cancel

Customizable display elements:

- **Home**: Displays the color of the current **Home position**.
- **Direction arrow**: Displays the color of arrows that indicate the direction of a chain geometry.
- **Point**: Displays the color of points during chain selection, e.g. the start point of an element.
- **Numbers**: Displays the color of numbers used to identify chain geometries.

- **Section**: Displays the color of the section geometries.
- **Clamp**: Displays the color of the clamp in SolidCAM-Turning.
- **Show tool**: Displays the color of the tool shown from the Job screen.
- **Pick highlight**: Selected elements, e.g. surfaces, solids, lines, etc. are highlighted using this color.
- **Show chain/3D model/Drill**: Displays the color of geometries when you use the Geometry>Show commands.
- **Chain Accept highlight**: Displays the color of chains that have been added to the currently active geometry.
- **General chain selection**: Color used to highlight entities suggested as consequent elements of the chain in auto select mode.
- **Limit/Working area**: Displays the color of the Limit geometry and Working area geometry.
- **Material boundaries**: Displays the color of the **Material boundaries**.
- **Vertical Type Def. Depth**: Displays the color of numbers used to identify the depth of the vertical type variable profile depth.
- **Sloped Type Def. Depth**: Displays the color of numbers used to identify the depth of the sloped type variable profile depth.

Load default color set

This option enables you to load the default colors of SolidCAM.

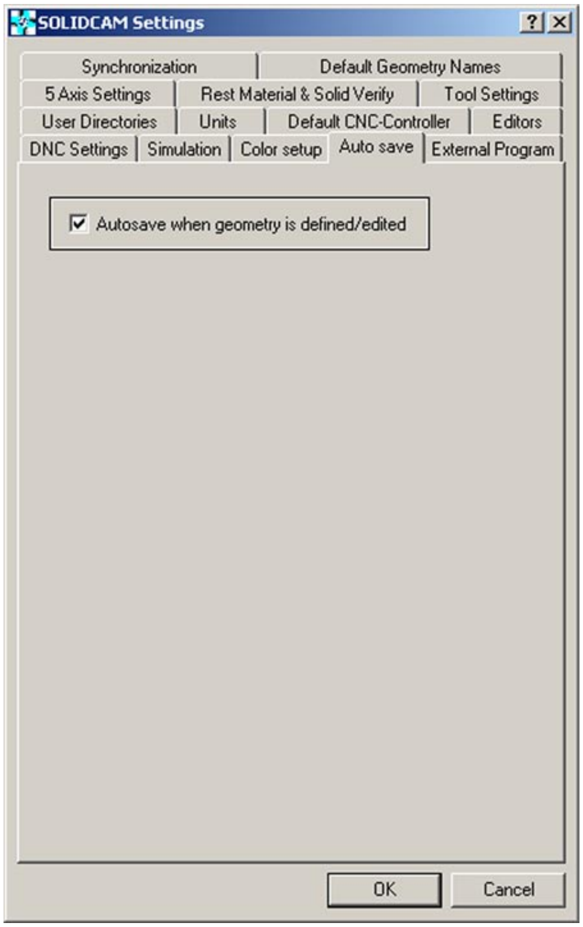
Host CAD background color

You can load a pre-defined set of colors for the customizable display elements. These colors have been selected by SolidCAM for the standard background colors of the Host CAD systems used by SolidCAM.

- Choose the background color of the Host CAD from the list.

The **Auto adjust** option enables you to load the optimal colors for the background of your Host CAD.

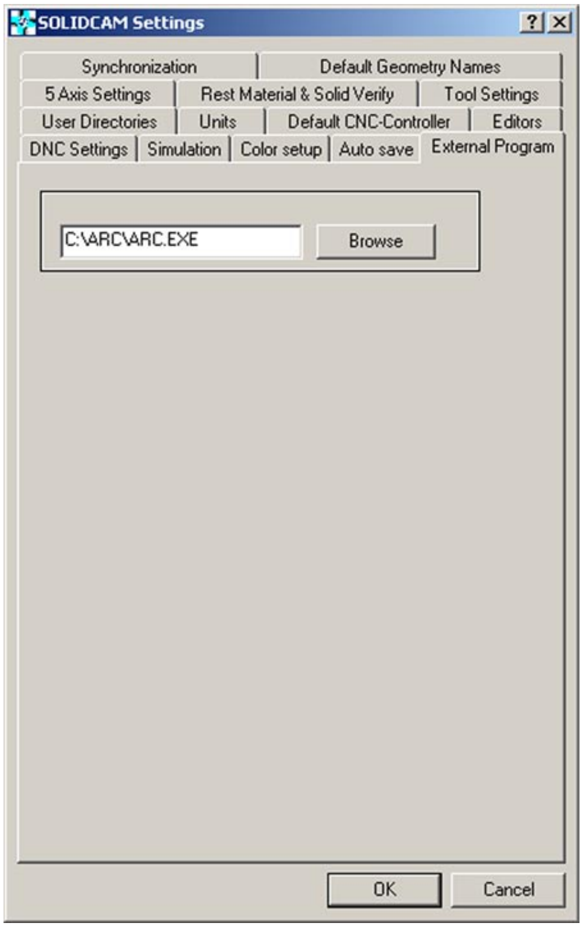
10.6 Auto-Save settings


SOLIDCAM Settings
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
DNC Settings
Simulation
Color setup
Auto save
External Program
OK
Cancel

☑ Autosave when geometry is defined/edited

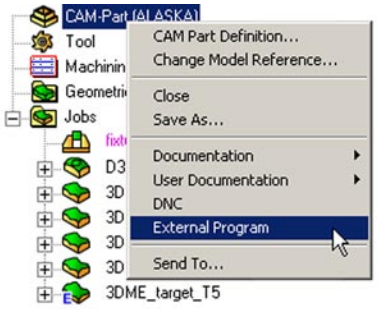
SolidCAM enables you to save your CAM-Part automatically after a geometry is defined or edited.

10.7 External program settings


SOLIDCAM Settings
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
DNC Settings
Simulation
Color setup
Auto save
External Program
Browse
OK
Cancel

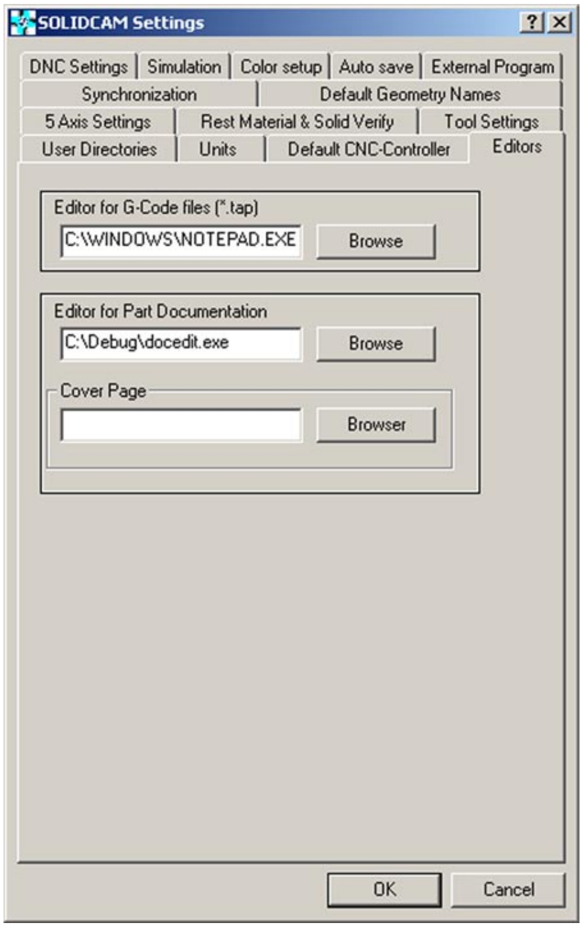
C:\ARC\ARC.EXE

SolidCAM enables you to define a path to an external program (Windows application or batch file) that can be loaded from the SolidCAM menu.


CAM-Part (ALASKA)
Tool
Machinin
Geometri
Jobs
fixt
D3
3D
3D
3D
3D
3DME_target_T5
CAM Part Definition...
Change Model Reference...
Close
Save As...
Documentation
User Documentation
DNC
External Program
Send To...

10.8 Editors Settings

The **Editors** register enables you to specify standard editors for text files created by SolidCAM.


SOLIDCAM Settings
DNC Settings
Simulation
Color setup
Auto save
External Program
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
Browse
Browse
Browser
OK
Cancel

Editor for G-Code files (*.tap): C:\WINDOWS\NOTEPAD.EXE

Editor for Part Documentation: C:\Debug\docedit.exe

Cover Page:

Editor for G-Code files (*.tap)

Specify the Editor to view, open and print generated G-Code files.

- Enter the path to a text editor or use **Browse** to select an application on your system.

Editor for CAM-Part Documentation (*.doc) files

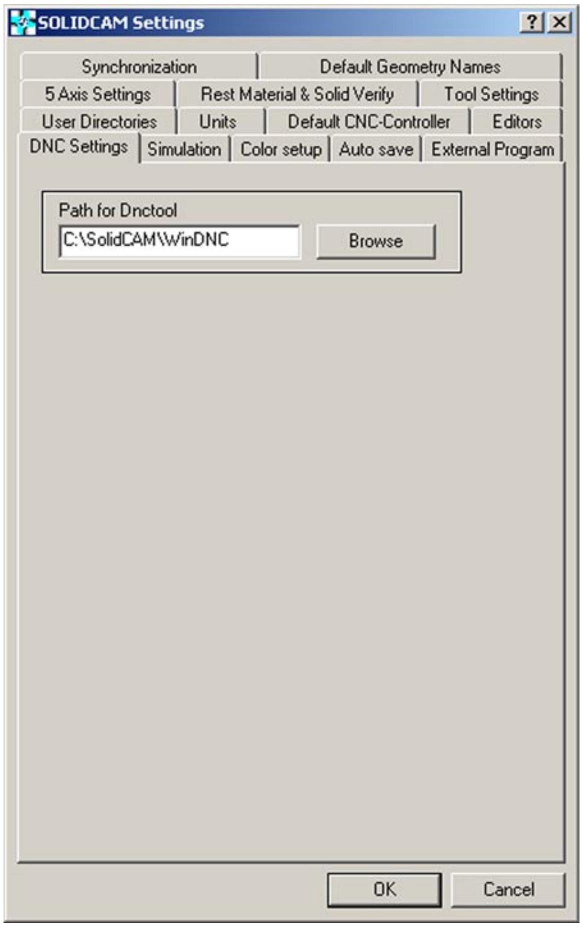
The documentation for CAM Parts will be generated in text files with the *.doc extension. As the default editor, the SolidCAM viewer.exe can view and print documentation files. You can, however, specify any text editor or word processor that can read *.doc files as the standard editor for your documentation.

- Enter the path to a text editor or use **Browse** to select an application on your system.

Cover Page

This option enables you to define the path to the Cover Page used for the **CAM-Part Documentation.**

10.9 DNC settings

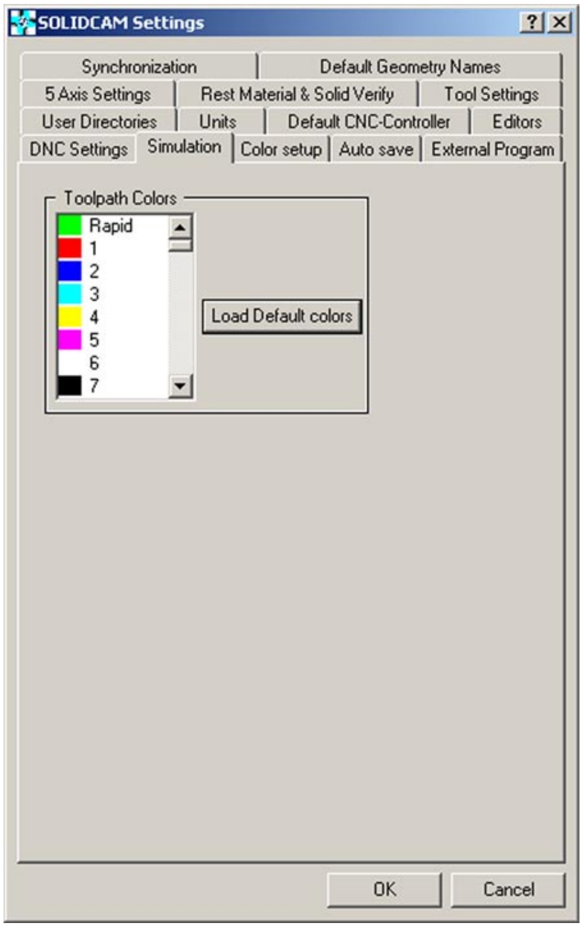

SOLIDCAM Settings
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
DNC Settings
Simulation
Color setup
Auto save
External Program
Browse
OK
Cancel

Path for Dnctool: C:\SolidCAM\WinDNC

This option enables you to define the path for the data transmitting software - the DNC program.

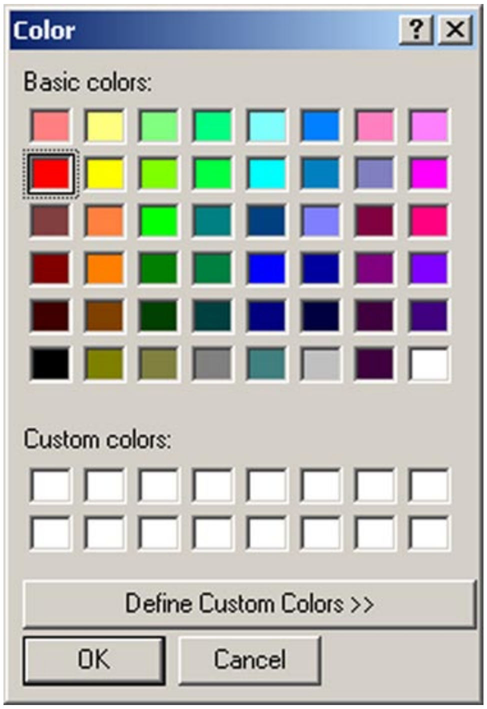
10.10 Simulation Settings

The **Tool Simulation Color** register enables you to customize the color of the tools used during the **Turning simulation** mode.


SOLIDCAM Settings
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
DNC Settings
Simulation
Color setup
Auto save
External Program
Toolpath Colors
Rapid
1
2
3
4
5
6
7
Load Default colors
OK
Cancel

For each tool number you can define a separate color. The colors selected in this dialog will be used for tools of current and future parts.

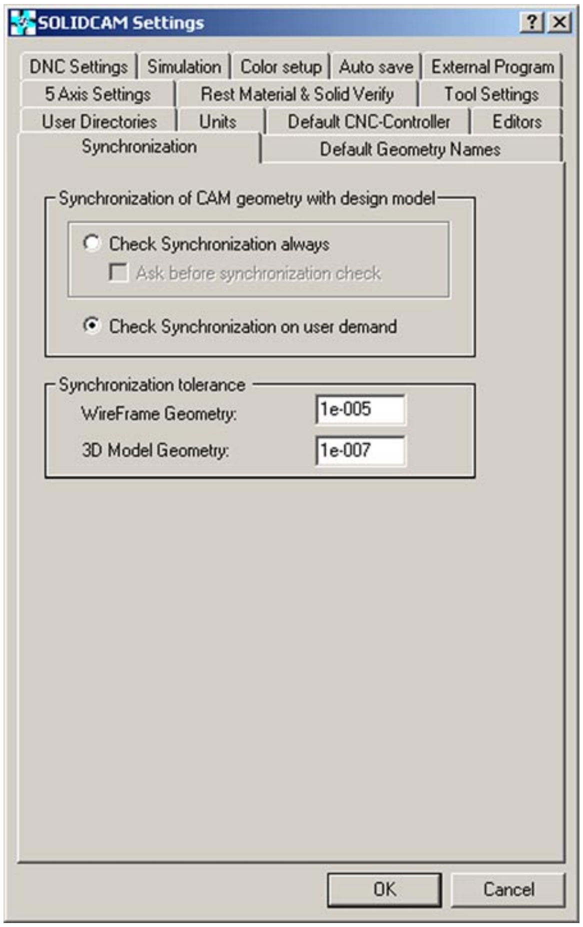
To change a color, click on the color symbol next to the tool number in the list. The standard Windows Color selection dialog will be displayed, enabling you to pick the color you want.


Color
Basic colors:
Custom colors:
Define Custom Colors >>
OK
Cancel

Load Default colors

Click on the button to load the default SolidCAM color settings for simulation.

10.11 Synchronization settings


SOLIDCAM Settings
DNC Settings
Simulation
Color setup
Auto save
External Program
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
Synchronization
Default Geometry Names
OK
Cancel

Synchronization of CAM geometry with design model

○ Check Synchronization always

☐ Ask before synchronization check

● Check Synchronization on user demand

Synchronization tolerance

WireFrame Geometry: 1e-005

3D Model Geometry: 1e-007

Check Synchronization always

Ask before synchronization check:

- **Marked**

  After every change in the part or in the assembly and rebuild, SolidCAM will prompt you before checking the synchronization of the geometries with the model or sketches.

- **Unmarked**

  The synchronization checking will be performed without any message or question.

Check synchronization on user demand

In this case, SolidCAM will not check the synchronization automatically. The **Check Synchronization** item in the **Job** and **Geometry** menus is enabled and you can choose it to start the synchronization.

Synchronization tolerance

These parameters define the value of the synchronization tolerance. During the synchronization check, SolidCAM compares the SolidCAM geometry and the SolidWorks model. If the difference between them is out of the specified tolerance (the SolidWorks model was updated); you have to synchronize the geometry with the updated model.

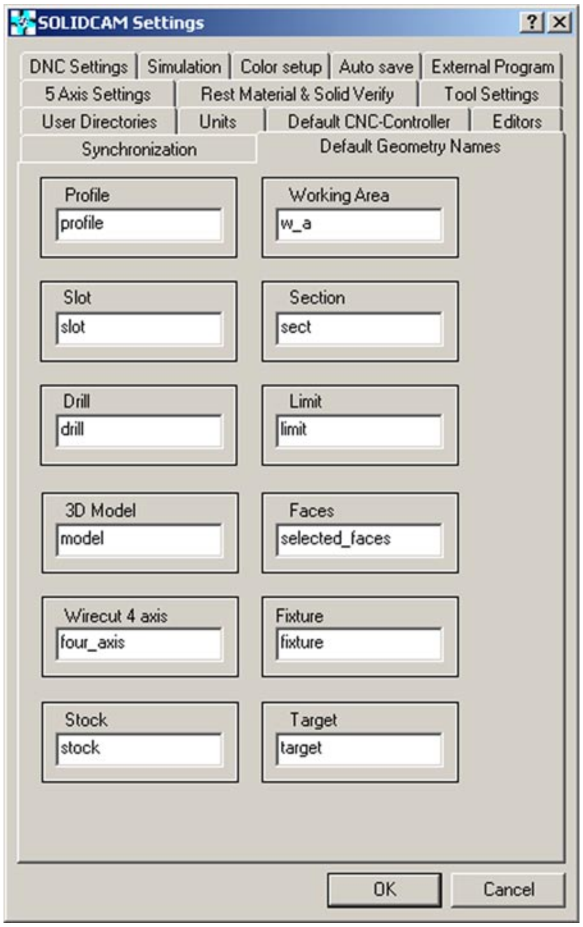
- **Wireframe Geometry**

  The value of the tolerance for the Wireframe geometry.

- **3D Model Geometry**

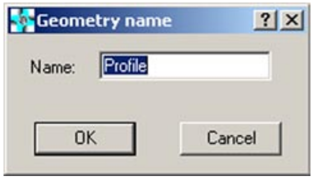
  The value of the tolerance for the 3D Model geometry.

10.12 Default Geometry Names


SOLIDCAM Settings
DNC Settings
Simulation
Color setup
Auto save
External Program
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
User Directories
Units
Default CNC-Controller
Editors
Synchronization
Default Geometry Names
OK
Cancel

Profile: profile
Working Area: w_a
Slot: slot
Section: sect
Drill: drill
Limit: limit
3D Model: model
Faces: selected_faces
Wirecut 4 axis: four_axis
Fixture: fixture
Stock: stock
Target: target

SolidCAM enables you to define and use default names for all geometry types. The **Default Geometry Names** tab was added to the **SolidCAM Settings dialog**.

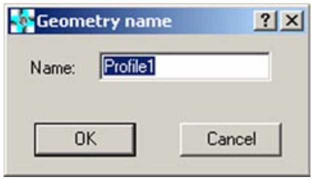

Geometry name
OK
Cancel

Name: Profile

This dialog enables you to define the **Default names** for SolidCAM geometries.

When a new geometry is defined, the default name will be displayed in the **Geometry Name dialog**. You can either confirm it or change it.

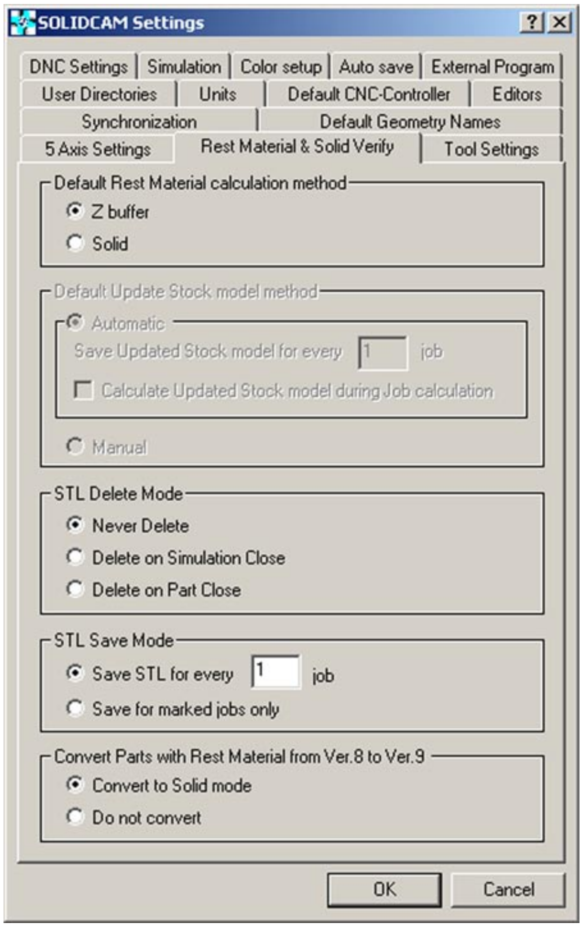
If the default name is already used for a previously defined geometry, SolidCAM offers you the default name with a next sequential number.


Geometry name
OK
Cancel

Name: Profile1

10.13 Rest Materal & SolidVerify Settings

SolidCAM enables you define the default settings for Rest Material calculation.


SOLIDCAM Settings
DNC Settings
Simulation
Color setup
Auto save
External Program
User Directories
Units
Default CNC-Controller
Editors
Synchronization
Default Geometry Names
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
OK
Cancel

Default Rest Material calculation method
● Z buffer
○ Solid

Default Update Stock model method
● Automatic
Save Updated Stock model for every 1 job
☐ Calculate Updated Stock model during Job calculation
○ Manual

STL Delete Mode
● Never Delete
○ Delete on Simulation Close
○ Delete on Part Close

STL Save Mode
● Save STL for every 1 job
○ Save for marked jobs only

Convert Parts with Rest Material from Ver.8 to Ver.9
● Convert to Solid mode
○ Do not convert

Default Rest Material calculation method

This option enables you to set the default method for the Rest material calculation.

- **Z buffer**

  In this mode, the rest material is calculated only for one machining direction.

- **Solid**

  This mode enables you to calculate the Rest material from all sides.

Default Update Stock model method

This option enables you to set the default method for the Stock model updating.

This option is active only with **Solid method** of the Rest material calculation.

- **Automatic method**

  This option causes SolidCAM to automatically calculate and save the rest material after each job. During the **SolidVerify simulation** of the separate jobs, SolidCAM updates the stock with the previously generated and saved rest material and performs the simulation on the updated model. Since the rest material models are saved after each job calculation, SolidCAM enables you to simulate any job for multi-sided CAM-Parts.

- **Manual method**

  SolidCAM enables you to manually save the Rest Material for a specific job during the **SolidVerify simulation**. The saved model can be used to update the stock. The simulation of the next jobs can be performed on the updated model.

STL Save mode

This option enables SolidCAM to save the **STL** file containing the Updated Stock model when the **Initialization of machined stock** option is active. This option is active either when the **Update Rest Material** option is not chosen or when the **Z-Buffer** mode is chosen.

The saved STL file will be used in SolidVerify to increase the simulation speed.

The following options are available:

- **Save STL for every N job**
- **Save for marked job only**

STL Delete mode

This option enables you to delete the STL files created by the previous option.

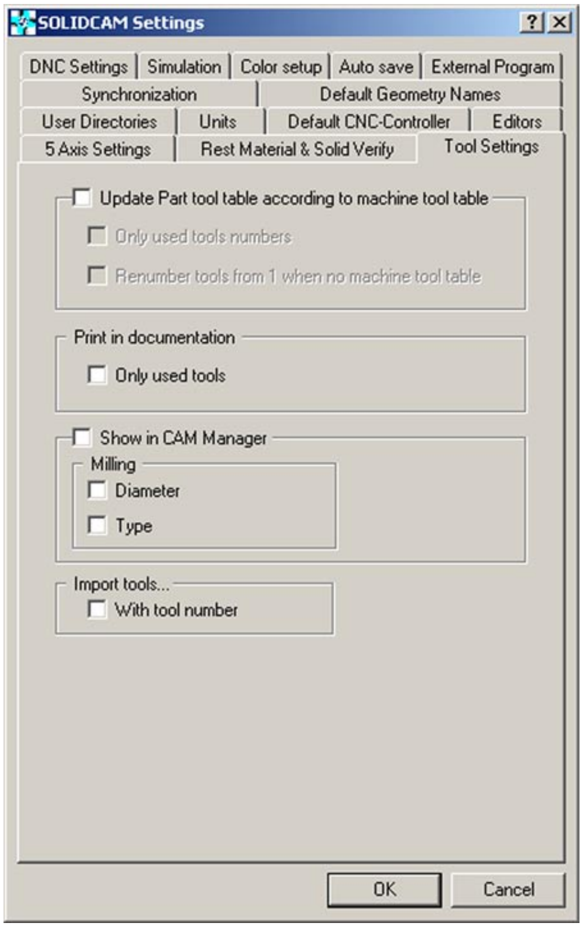
- **Never Delete** – the STL files will not be deleted.
- **Delete on Simulation Close** – the STL files will be deleted when the Simulation is closed.
- **Delete on Part Close** – the STL files will be deleted when the CAM-Part is closed.

Convert parts with Rest Material from Ver. 8 to Ver. 9

This option enables you to define how the CAM-Part with the Rest Material will be converted.

- **Convert to Solid mode** – the CAM-Part will be converted to the Solid mode of the Stock Model Update.
- **Do not convert** – the part will not be converted; the Z-Buffer mode will be used.

10.14 Tool Settings


SOLIDCAM Settings
DNC Settings
Simulation
Color setup
Auto save
External Program
Synchronization
Default Geometry Names
User Directories
Units
Default CNC-Controller
Editors
5 Axis Settings
Rest Material & Solid Verify
Tool Settings
OK
Cancel

☐ Update Part tool table according to machine tool table
☐ Only used tools numbers
☐ Renumber tools from 1 when no machine tool table

Print in documentation
☐ Only used tools

☐ Show in CAM Manager
Milling
☐ Diameter
☐ Type

Import tools...
☐ With tool number

Update Part tool table according to Machine Tool Table

By setting this option the user instructs SolidCAM to automatically update the Part Tool Table with tools from the Machine tool table defined in the MAC–file, when the CNC-controller for the CAM-Part is changed.

Only used tools numbers

This option enables you to update only those tools that were used in jobs. Tools that were not used will be removed from the Part Tool Table. Used tools that were not found in the Machine Tool Table will be numbered with the next sequential numbers.

- **Renumber tools from 1 when no machine tool table**

  The option removes all unused tools from the Part Tool Table. The used tools will be renumbered starting from number 1. If the **Renumber tools from 1 when no Machine tool table** option is not active, SolidCAM removes all unused tools from the Part Tool Table. The used tools will not be renumbered.

This option is available only if the **Only used tools numbers** option is activated and only if the Machine Tool Table is not defined for the MAC of the new CNC-controller.

Print in documentation - Only used tools.

This option enables SolidCAM to include in the CAM-Part documentation only the tools from the Part Tool Table used in jobs.

Show in CAM Manager

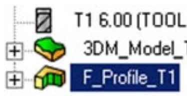
This option enables you to display the tool headers in the CAM-Manager, whenever a new tool is used. The Tool Header is displayed before the job where the new tool is used.

Diameter

This option displays the tool diameter.

Type

This option displays the tool type.



T1 6.00 (TOOL END MILL)
3DM_Model_T1
F_Profile_T1

Imprt tools

With tool number

This option enables you to import tools to the Part Tool table with their tool numbers. If this option is not checked, SolidCAM will assign a first free number to the imported tool.

Index

Symbols

2D simulation 394
3D Drill Job 387
3D Milling 309
3D Model Geometry 156
3D Simulation mode 406

A

Adding a Job 196
Adjacent tool path connection 264
Approach 246, 272
Approximation 434
Arc approximation 353, 373, 379
Auto-Save settings 440
Automatic Sort 218
Axis type 71

B

Boring 283

C

Calculate 213
CAM-Part 27
CAM-Part data dialog 28
CAM-Part Documentation 81
CAM-Part Lower level 53
CAM-Part name 29
CAM-Part Upper level 53
Chain 169
Chains direction 181
Chains sorting 183
Chain Direction 173
Chain Options dialog 172
Chain selection 172
Chain selection dialog 179
Chain selection settings 434
Chamfer 237
Check faces 318
Check Synchronization always 445
Check synchronization on user demand 445
Choose Faces dialog 318
Circular Pocket 368
Clean flat 325
Clearance level 53
Clear offset 233
CNC-Controller 71
CNC-controllers settings 437

Color Setup 438
Compare target and machined stock 415
Compensation 244, 270
Connection to CAD system 17
Connect islands 260
Connect type 300
Constant-Z 370
Constant-Z flat floor machining 377
Constant-Z Tolerance 435
Constant-Z wall machining 371
Constraint 297
Contour Data dialog 258
Contour Roughing 341
Cross finish 354
Current Tool Table 89
Cutting direction 256, 366

D

Default CNC-controller settings 436
Default Geometry Names 447
Default Rest Material calculation method 448
Default tool data 115
Default Update Stock model method 448
Depth 204
Descent 328, 346
Descent Angle 299, 302
Diameter Offset Number 116
Directory 29
DNC settings 443
Documentation Editor 82
Dongle Update 21
Down Step 232, 251
Down step 325, 385
Drilling 282
Drill cycle type 281, 389
Drill depth 280
Drill Geometry 150
Drill Job 279
Drill points 333
Drive faces 318

E

Editors Settings 442
Engraving Data 301
Engraving depth 306
Engraving geometry 307
Engraving Job 382
Engraving Limit 306
Engraving technology 384
Equal distance 357
Exit Material 257
Export all tools to Excel 107

Extend to edge 357
External program settings 441

F

Feed 116
Feed and Speed default 142
Fillet size for last cut 234
Finish cycle 300
Fixture 225
Fixture dialog 226
Floor Offset 270, 327

G

G-Code 426
Gap maximum 435
Gap minimum 435
Getting Help 25
Global holders table 114
Global shape-tools table 113

H

Hatch Data dialog 253
Hatch Roughing 340
Hidden line mode 413
Home 31
Home operations 56
Host CAD simulation mode 404
Hybrid mode 412

I

Import all tools from Excel 107
Installing SolidCAM 14
IPX Protocol 20
Island Offset 270

J

Job Group 214
Job Interface 197
Job Lower level 204
Job name 203
Job Sequence 216
Job Upper level 204

L

Length Offset number 116
License Server 20
Limits 302
Limit Data 303

Limit Geometry 167
Linear finish 350
Local holders table 114
Local shape-tools table 113

M

Machine Tool Table 89
Machining area 304
Mac home number 33
Mac Options 69
Maximum down step 372
Message 209
Metric/Inch 433
Milling Levels 204
Minimal clean condition 269
Minimum down step 372
Mirror/Rotate point 208
Model Box 35
Model name 29
Model Reference 79
Multi-chain 169
Multi-Sided Home 31

N

Network License Installation 19

O

Offset cutting 356
Offset from model 389
Offset geometry 357
Offset on profile 232
Offset on surface 299
Open Pocket machining 334
Overlap 270, 299, 302, 325

P

Part Tool Table 89
Path 431
Peck 282
Pencil milling 380
Permanent tool 105
Plunging Pattern 268
Plunging pattern 342
Pocket Job 250
Pocket Types 252
Position 33
Post-processor files directory 436
Profile/Pocket Geometry 161
Profile Direction 246
Profile extension 246

Profile Job 230
Program Folder 16
Program Number 75
Projections Mode 46

R

Radius compensation 244
Render mode 412
Rest Materal & SolidVerify Settings 448
Rest Material 237, 337
Rest material 59
Rest Material simulation mode 420
Retreat 246, 276
Roughing 324
Rough cycle 300

S

Scallop 299, 301, 372, 378
Section depth type 289
Section Geometry 165
Semi-Finish/Finish 343
Semi-Finish/Finish strategies 349
Sequence number 99
Settings 430
Setup Language 14
Setup mode 400
Shape-Tools dialog 131
Shaped Tools 130
Shaped Tool segments 136
Show/Hide Gouges 414
Show/Hide Split Solids 414
Simulation 392
Simulation Controls 393
Simulation menu 417
Simulation Mode 393, 400
Simulation Settings 444
Simulation speed 405
Simulation Toolbar 399
Single License Installation 14
Slot extension 289
Slot Geometry 164
Slot Job 286
Slot levels 289
Slot with constant depth 286
Slot with variable depth 286
SolidCAM Help 25
SolidCAM Manager 23
SolidVerify simulation mode 409
Spin 116
Spiral Finish 361
Spline approximation 171
Starting SolidCAM 22

STL 417
STL Delete mode 449
STL Save mode 449
Stock model 58
Subroutine Number 75
Surface Curvature 323
Surface Data 298
Surface offset 301, 326
Surface tolerance 322
Synchronization 187
Synchronization check 188
Synchronization Settings 445
Synchronization Tolerance 446
System requirements 14

T

Tapping 283
Target model 59
TCP/IP Protocol 19
Tolerance 321, 383
Tools Filter 98
Tool Curvature 323
Tool data 108
Tool holders dialog 118
Tool Holder segments 124
Tool holding system 117
Tool Length 112
Tool Material 115
Tool number 99
Tool on working area 314
Tool options dialog 67
Tool path connections 360
Tool path tolerance 323
Tool Range 99
Tool Settings 450
Tool Side 243
Tool start level 53
Tool Tables 89
Tool Table dialog 97
Tool Types 86
Tool Z Level 54
Topology page 110
Translated Surface Job 293
Trochoidal milling 249

U

Units Settings 433
Update to Machine Tool table 106
Updating Stock Model 65
User Directories Settings 431
User Directory for SolidCAM-Parts 431
User Directory for SolidCAM-Tables 432

V

VerifyPlus simulation 399

W

Wall finish 271
Wall Offset 270, 327
Wireframe Geometry 160
Wire mode 413
Working Area 162, 311
Working area dialog 312
Work Material 70

Z

Zero Compensation 245
Z Buffer 65
Z buffer 448

Document Number: **SCMUGENG05001**